Assessment of Turbulence Models on a Backward Facing Step Flow Using OpenFOAM®

Anugya Singh, Aravind S, Srinadhi K and Kannan B T

Department of Aerospace Engineering, SRM Institute of Science and Technology, Kattankulathur, Chennai, Tamil Nadu, India

Email: skyinventorbtt@gmail.com; btk@alumni.iitm.ac.in

Abstract. Studies on turbulent inlet flows have a wide variety of applications, Backward Facing Step (BFS) facilitates this study. The phenomena of flow separation occur due to an abrupt change in geometry, causing the creation of a re-circulation zone and a flow re-attachment point. The present numerical study focuses on the assessment of different turbulence models. The flow velocity is in the turbulent region characterized by the Re=7000. Steady State Reynolds Averaged Navier Stokes equations are solved along with various turbulence models using OpenFOAM® which is an open-source Computational Continuum Mechanics toolbox for conducting numerical simulations. The simulated data is processed using Paraview and the assessment of the turbulence models is performed via quantitative (re-attachment length) and qualitative (Line Integral Convolution visualizations) methods.

1. Introduction
The Backward Facing Step (BFS) is a model that has applications in multi-disciplinary domains. This model helps to understand the changes in the flow property arising due to its separation. The geometry and conditions are visible in many cases related to aerodynamics such as industrial aerodynamics which includes flow around the vehicles and buildings. The phenomenon can be observed with aircraft aerodynamics as well which includes the flow separation and reattachment over an airfoil at different angles of attack. This model incorporates the most important types of flow features that are free shear flow separation, vortex evolution and the re-attachment. Along with these applications, the flow dynamics, various parametric studies, chemical and mechanical characteristics with respect to the theoretical, experimental and computational aspects were discussed in the subsonic regime and an emphasis was made on the fluid mixing process, perturbation and future trends in the supersonic regime along with heat transfer as well in the review by Chen et.al. [1].

The study of these features has already been done by several researchers and the data has been extracted, not just experimentally but also numerically. With upcoming new software and numerical codes, the study of this model has been done using several ways so that the numerical and experimental data can be compared and the differences can be understood.

To study such a phenomenon, many turbulence models have been created and each model is designed differently. But it is crucial to decide which turbulence model works the best for these types of problem and hence many researchers came up with comparison ideas and started working on this problem. One of such researchers, Thangam [3] worked on the two-equation k-ԑ model and modified it so that it could incorporate the imbalance between turbulence dissipation and production, he also worked on LES method with eddy-viscosity representation in order to check the results and compared them with the experimental results by Eaton and Johnston [2] and found that the modified model and LES could accurately predict the major flow features.

Haque et.al. [4] carried out the analysis for two and three-dimensional models for lower Reynolds Number that is 600 and 1000. These gave separate results for 2D and 3D cases and it was found that
this was due to the difference in mass flow rate values. Also, the SST turbulence model was seen to be more accurate than the epsilon-based turbulence models. Detached Eddy Simulation (DES) method was also used and it was found that it accurately predicted the three-dimensional vortical structure when compared to RANS. Nor Azwadi C. Sidik et.al. [5] worked on the comparison between Lattice Boltzmann Equation (LBE) - Large Eddy Simulation (LES) at a Reynolds number of 5100 and compared these results to Navier Stokes Direct Numerical Simulation (NSDNS). The model created had an expansion ratio of 1.2 with length and breadth dimensions as 30h and 6h respectively, where ‘h’ was the step height. On comparing the results, it was found that the results obtained were almost the same and hence it was concluded that this model could work at par with other computational methods.

Jehad et.al. [6] carried out another set of two-dimensional simulations in which a comparison was carried out between Standard k-epsilon, realizable k-epsilon, and k-omega SST for a model with expansion ratio 1:3, at Reynolds number 13200. The re-attachment lengths on comparison showed that the k-omega SST had approximately the same result as the experimental value but the other two under predicted the results. Araujo et.al. [7] also worked on a two-dimensional model having an expansion ratio of 2, for studying flow over a Backward-Facing Step. The k-epsilon, k-omega, k-omega SST and Reynolds Stress Model (RSM) which is a second-order closure model was used to check the accuracy. It was observed that k-omega SST gave the most accurate results whereas the RSM model came to a good agreement with the turbulence intensity profile when compared to other models. Armaly et. al. [8] investigated a BFS flow in theoretical, experimental and computational aspects respectively. The Reynolds number was varied from 70 to 8000, hence it covered laminar, transition and turbulent regimes. The conclusion was derived after comparison of data from all the three aspects that the variations in the separation length with Reynolds number caused the characterization of different flow regimes.

An investigation by Kannan et.al. [13] for the choice of turbulence models in the case of axisymmetric jet flow, provided better insights with the comparative results by examining different flow fields and parameters that represent its physics. It was found that results from the first-order standard k-epsilon and Realizable k-epsilon model were better than the second-order Reynolds Stress Transport Model. A similar assessment is needed for the internal flow for the combustion applications. The present work will address these in the form of Cold Flow analysis to provide a better understanding of the flow physics and phenomena, which could be used in the later stages to aptly simulate hot flow analysis using air-fuel mixture.

These analyses and its results discussed above were mostly carried out in ANSYS FLUENT®, but in order to understand the same Backward Facing Step problem in-depth, our work was successfully carried out and completed using the open-source software named OpenFOAM®. This software was chosen as it is designed such that one can modify the internal equations easily by making changes in the script, hence carrying out simulations as per the requirements of the experiment. It can carry out parallel processing in large processor arrays, reducing the overall calculation time which gives it an added ability to solve complex geometries as well [12].

The model for these simulations was created and analyzed for a single turbulence model, along with a grid independence study by Kumar et. al. [14]. The present work is focused on using several turbulent models including both Linear and Non-Linear eddy viscosity turbulence models, and analyzing its effect on the flow inside the geometry.

2. Computational Methodology

2.1 Geometry & Meshing

The geometry was modeled using the ‘blockMesh’ dictionary file in OpenFOAM®. It was modeled to be two dimensional in nature with unit width. The design of the model was based on the experimental model taken from Armaly et.al. [8], hence the aspect ratio of 1:18 and the expansion ratio (H/h) of 1.94 (where H=10.1mm & h=5.2mm) were modeled as shown in figure 1.
Figure 1. The geometry of the Backward Facing Step Model created in OpenFOAM®.

For ease of meshing, 6 divisions were made out of the model and the following names were provided to the faces: inlet (extreme left face), outlet (extreme right face), front & back (in the plane of paper/screen), upper-wall (total upper surface), and lower-wall (total bottom surface). The mesh was provided with biasing such that the density of mesh was more towards the middle section of the model. The grid independence study was conducted for coarse, medium and fine mesh and the medium was chosen for computational efficiency [14]. The mesh created was as shown in figure 2.

Figure 2. The medium-mesh of the Backward Facing Step Model created in OpenFOAM®.

2.2 Governing Equations

Since the aim of this work is to analyze the numerical models in the open-source software OpenFOAM®, all the governing equations mentioned here are the ones used in the given software. This software has simplified the main equations further for obtaining the required results. The incompressible Reynolds Averaged Navier-Stokes equation governs the flow on the Backward Facing Step geometry. The momentum equation for solving RANS problems in OpenFOAM® is given by equation (1) [10]:

$$\frac{\partial \bar{u}_i}{\partial x_j} \left[ \nu_{eff} \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \right] = -\frac{\partial p}{\partial x_i}$$  \hspace{1cm} (1)

The Standard k-epsilon model is the basic model for calculating incompressible flows. It is said to have several features such as it over predicts the k value at stagnation points and hence needs near-wall treatment. In the case of OpenFOAM®, the right-hand side of the equation is solved such that it contains Rapid Distortion Theory (RDT) and buoyancy is not taken into account. It contains two steady-state equations which include turbulence kinetic energy k and turbulence dissipation rate $\varepsilon$.

The equations for each of them are given respectively as follows [11]:

$$\frac{D}{Dt} (\rho k) = \nabla. (\rho D_u \nabla k) + \sigma_k - \frac{2}{3} \rho (\nabla. u) k - \rho \varepsilon + S_k$$  \hspace{1cm} (2)

$$\frac{D}{Dt} (\rho \varepsilon) = \nabla. (\rho D_\varepsilon \nabla \varepsilon) + \frac{C_1 G_k \varepsilon}{k} - \left( \frac{2}{3} C_4 - C_{3,RDT} \right) \rho (\nabla. u) \varepsilon - C_2 \rho \frac{\varepsilon^2}{k} + S_z$$  \hspace{1cm} (3)

Here for initialization of isotropic turbulence, turbulence kinetic energy and dissipation rate are estimated using the formulae

$$k = \frac{3}{2} \left( \left[ u_{ref} \right] \right)^2$$  \hspace{1cm} (4)
Where, $C_\mu = 0.09$, $C_1 = 1.44$, $C_2 = 1.92$, $C_3, RDT = 0$.

The Realizable k-epsilon model contains two steady-state equations for turbulence calculations which are given as follows [11]

\[
\frac{D}{Dt} (\rho k) = \nabla \cdot (\rho D_k \nabla k) + \rho G - \frac{2}{3} \rho (\nabla \cdot u)k - \rho \varepsilon + S_k
\]

\[
\frac{D}{Dt} (\rho \varepsilon) = \nabla \cdot (\rho D_\varepsilon \nabla \varepsilon) + C_1 \rho |S| \varepsilon - C_2 \rho \frac{\varepsilon^2}{k} + (\frac{\varepsilon}{\bar{\varepsilon}})^{1.5} k + S_\varepsilon
\]

Where turbulence viscosity and $C_\mu$ are given by

\[
v_\tau = \frac{C_\mu k^2}{\varepsilon}
\]

\[
C_\mu = \frac{1}{A_0 + A_s U^s_z}
\]

The Standard k-omega model is similar to the Standard k-Epsilon model where the turbulence kinetic energy remains the same but this model uses specific dissipation which is given by $\omega = \frac{\varepsilon}{k}$ [10] hence leading to the omega equation.

The k-omega SST model was created in order to eliminate the dependency on the freestream values obtained from k and omega. It can successfully capture the flow separation, better than the k-omega model. This numerical model contains two steady-state equations for turbulence specific dissipation rate and turbulence kinetic energy which is given as follows [11]:

\[
\frac{D}{Dt} (\rho \omega) = \nabla \cdot (\rho D_\omega \nabla \omega) + \frac{\rho \gamma G}{\nu} - \frac{2}{3} \rho \gamma \omega (\nabla \cdot u) - \rho \beta \omega^2 - \rho (F_1 - 1) CD_{\varepsilon \omega} + S_\omega
\]

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

Here turbulence Viscosity is,

\[
v_\tau = \frac{k}{a_1 \max (a_3 \omega, b_1 F_{23} S)}
\]

Here for initialization of isotropic turbulence, turbulence kinetic energy and turbulence specific dissipation rate is estimated using the formulae

\[
k = \frac{3}{2} (l[l u_{re}])^2
\]

\[
\omega = \frac{k^{0.5}}{C_\mu L}
\]

Where $C_\mu = 0.09$, $a_1 = 0.31$, $b_1 = 1.0$, $\beta^* = 0.09$

The Shih Quadratic k-epsilon model contains two steady-state equations for turbulence calculations which are given as follows [12]:

\[
\frac{D}{Dt} (k) = \nabla \cdot (D_k \nabla k) + G - \varepsilon
\]
Here $G$ is turbulence generation which is expressed as,

$$
\frac{D}{Dt}(\varepsilon) = \nabla \cdot (D_\varepsilon \nabla \varepsilon) + C_4 G \frac{\varepsilon}{k} - C_2 \varepsilon^2 \frac{\varepsilon}{k}
$$

(16)

The Lien Cubic k-epsilon model contains two steady-state equations for turbulence calculations which are given as follows [11]:

$$
\frac{D}{Dt}(k) = \nabla \cdot (D_k \nabla k) + G - \varepsilon
$$

(18)

$$
\frac{D}{Dt}(\varepsilon) = \nabla \cdot (D_\varepsilon \nabla \varepsilon) + C_{\varepsilon 1} G \frac{\varepsilon}{k} - C_{\varepsilon 2} \varepsilon^2 k + E
$$

(19)

2.3 Boundary Conditions

The analysis was run for Reynolds Number of 7000 that is on calculating, the velocity provided for was 9.89 m/s. The boundary conditions provided for the analysis were given as shown in table 1.

| Position                  | OpenFOAM® Boundary Condition |
|---------------------------|------------------------------|
| Inlet Face                | Uniform Constant Value       |
| Outlet-Face               | Zero Gradient                |
| Lower & Upper Wall Faces  | Wall (no-slip)               |
| Front & Back Faces        | Empty                        |

Here, the zero-gradient refers to the condition in which the gradient value is provided as zero in the perpendicular direction of the patch. Wall is the boundary condition in which the velocity of the fluid is said to be zero near the wall. The inlet face was provided a constant velocity profile with a fixed value of 9.89 m/s in order to maintain the Reynolds number as 7000. To maintain a two-dimensional geometry, an empty boundary condition was applied so that the faces with unit width need not be considered.

3. Results and Discussions

The analysis that was performed for the six numerical models gave the reattachment lengths as shown in table 2. The data was extracted from the Coefficient of Friction values by plotting a graph against the non-dimensionalized distance that is $x/S$, as shown in figure 3. These values were calculated by finding that specific distance value at which the value of the Coefficient of Friction becomes zero after the step [9]. This value was then compared with the experimental reattachment length by Armalyet. al. [8].

| Numerical Model            | Re-attachment length (mm) | x/S |
|----------------------------|----------------------------|-----|
| k-omega SST                | 43.89                      | 8.96|
| k-Omega                    | 45.02                      | 9.18|
| Lien Cubic k-epsilon       | 35.51                      | 7.25|
| Shih Quadratic k-epsilon   | 32.77                      | 6.68|
| k-epsilon                  | 28.87                      | 5.89|
| Realizable k-epsilon       | 40.13                      | 8.19|
| Experimental [8]           | 39.40                      | 8.04|
On comparing the experimental data with the analysis data, it is clearly visible that the value obtained by Realizable k-epsilon is the closest to the experimental data and that none of the models could predict the exact re-attachment length for the given case. The models k-omega SST and k-Omega have over-predicted the reattachment lengths whereas, the models Lien Cubic k-epsilon, Shih Quadratic k-epsilon and k-epsilon have under-predicted the reattachment lengths where the numerical model k-epsilon has given the least value.

The visualizations that are presented in figure 4 represent the velocity magnitude (with Line Integral Convolution) plot in the x-direction of the Backward Facing Step model. It can be seen that two separated flows are seen right after the step, in the re-circulation zone. These zones have different sizes for different models which is very intriguing to see. The stagnation point location of both the primary and secondary vortices also differs for the different models. It can be observed that the secondary vortex is very small in size for k-epsilon, Shih Quadratic k-epsilon and Lien Cubic k-epsilon numerical models for the given Reynolds number, whereas in k-Omega and k-Omega SST the secondary vortex is of moderate size. Using these numerical models, the re-separation point and the re-attachment point that occurs between the two vortices can be comparatively easily studied than experimental methods.

Numerical model Realizable k-epsilon has the most differing image, as it has three vortices formed in the re-circulation zone hence consisting of three separate stagnation point, also the stagnation point for the primary vortex is seen to be close to the re-attachment zone of the main flow. These variations in flow vortices have probably led to the change in wall shear forces at the bottom wall and hence leading to the graph that is plotted for Cf. It can be observed in the graph as it shows three significant dips while gradually increasing to its maximum value.

The interactions between the primary and secondary vortex play an important role in predicting the flow and the reattachment lengths. These values can, therefore, be used to further study the prediction of the location of these stagnation points which can add to the prediction of flow and re-attachment length in the further stages with respect to time (unsteady analysis).
Figure 4. Flow re-attachment models for all the numerical models which include: (Top to bottom) k-epsilon, k-omega, k-omega SST, Lien Cubic k-epsilon, Realizable k-epsilon and Shih Quadratic k-epsilon.

4. Conclusions
The numerical simulations for Backward Facing Step flow were carried out successfully at the Reynolds number of 7000. The quantitative data was obtained by comparing the re-attachment lengths of different turbulence models using the Coefficient of Friction graph with respect to the experimental data. The numerical model Realizable k-epsilon gave the closest value quantitatively, whereas the numerical model k-omega and k-epsilon over predicted and under predicted the re-attachment lengths respectively. The position of stagnation point and the size of the recirculation have also been observed for separate models using Velocity Magnitude contours (with Line Integral Convolution) and k-Omega model was seen to have the biggest primary vortex hence leading to the maximum value for flow re-attachment length. The k-Omega SST model was observed with a moderate size of the secondary vortex as well as the primary vortex when compared with all other models. Thus, it can be said that it has provided solutions that are quantitatively and qualitatively closer to the experimental data.
5. References

[1] Chen L, Asai K, Nonomura T, Xi G and Liu T 2018 A review of Backward-Facing Step (BFS) flow mechanisms, heat transfer and control Thermal Science and Engineering Progress 6 194-216

[2] Eaton J and Johnston J.P 1980 Turbulent flow reattachment: an experimental study of the flow and structure behind a backward-facing step Technical Report MD-39 (Stanford University, CA)

[3] Thangam S 1992 Turbulent Flow Past the Backward-Facing Step: Modeling and Computations In Instability, Transition, and Turbulence 600-610 Springer New York NY

[4] Haque A, Ahmad F, Yamada S and Chaudhry S R 2007 Assessment of Turbulence Models for Turbulent Flow over Backward Facing Step Proceedings of the World Congress on Engineering 2 London U.K WCE

[5] Nor Azwadi C S, Ghaderian J and Sadeghipour S 2013 Simulation of Turbulent Flow Over a Backward Facing Step Using Lattice Boltzmann Method Journal of Engineering Science and Technology 8 670 – 782

[6] Jehad D G, Hashimb G A, Zarzoorc A K and Nor Azwadid C S 2015 Numerical Study of Turbulent Flow over Backward-Facing Step with Different Turbulence Models Journal of Advanced Research Design 4 20-27

[7] Araujo P P andRezende A L T 2017 Comparison of Turbulence Models in the Flow over a Backward-Facing Step International Journal of Engineering Research & Science 3

[8] Armaly B, J. Durst F, Jose P and Schönung B 1983 Experimental and Theoretical Investigation of Backward-Facing Step Flow Journal of Fluid Mechanics 127 473 - 496

[9] Kopera Michal K, Kerr Robert M, Hugh B and Dwight B 2017 Simulation of turbulent flow over a backward-facing step at Re=9000 Physics of Fluids in review

[10] Cappelli D and Mansour N N 2013 Performance of Reynolds Averaged Navier-Stokes Models in Predicting Separated Flows: Study of the Hump Flow Model Problem. 31st AIAA Applied Aerodynamics Conference, Fluid Dynamics and Co-located Conferences (San Diego, CA) (AIAA 2013-3154)

[11] OpenFOAM User Guide: https://www.openfoam.com/documentation/guides/latest/doc/guide-turbulence-ras-linear-eddy-viscosity-models.html

[12] Jones D A, Chapuis M, Liefvendahl M, Norrisson D and Widjaja R. RANS Simulations using OpenFOAM Software: Published by, (DST-Group-TR-3204) Maritime Platforms Division DST Group Defence Science and Technology Group 506 Lorimer St Fishermans Bend Victoria 3207 Australia

[13] Kannan B T, Karthikeyan S and Sudararaj S 2017 Comparison of Turbulence Models in Simulating Axisymmetric Jet Flow Innovative Design and Development Practices in Aerospace and Automotive Engineering Lecture Notes in Mechanical Engineering 401-407

[14] Kumar A S, Singh A and Kannan B T. 2020 Simulation of Backward Facing Step Flow Using OpenFOAM® AIP Conference Proceedings 2204 030002