Numerical Simulation of Fluid Flow over a Modified Backward-Facing Step using CFD

Himanshu Banait[1]*, Atharvasingh Bais[1], Kewal Khondekar[1], Rupesh Kumar Choudhary[1]
M. B. Bhambere[2]

[1]B.E student, Mechanical Engineering, Shri Sant Gajanan Maharaj College of Engineering, Maharashtra, India
[2]Assistant Prof., Mechanical Engineering, Shri Sant Gajanan Maharaj College of Engineering, Maharashtra, India

---***---

Abstract - *In this paper, numerical analysis is carried out on backward-facing step geometry in the Driver and Seegmiller experiment and then by modifying the backward-facing step geometry. Modified geometry will alter the size and characteristics of the recirculation vortex and turbulent kinetic energy profile downstream of the step. The application of sudden expansion geometry can be found in the combustor where the distribution of turbulent kinetic energy within the recirculation region determines the burning velocity of fresh reactants. Based on the CFD package ANSYS Fluent, we carried out the non-reactive numerical simulation. Numerical simulation was performed on 2-D geometry using the Reynolds Averaged Navier-Stokes (RANS) approach in the framework of the SST $k-\omega$ turbulence model. The experimental reference by Driver and Seegmiller was used for validation purposes. For the modified geometry, results show an increase in turbulent kinetic energy in the recirculation region and a slight decrease in reattachment length as compared to the original/traditional backward-facing step geometry.*

Key Words: *CFD; backward-facing step; Recirculation; Turbulent kinetic energy; SST k- ω*

1. Introduction

Flow over a backward-facing step is a classic fluid flow problem used to study turbulent separated-reattaching flows. Separated flow is generally observed in external aerodynamic and flow affected by an adverse pressure gradient. Turbulence plays a major role in flow separation and the effect of such a phenomenon on fluid flow is difficult to predict in complex geometry. Due to the simplicity of the geometry and availability of a large number of experimental results, flow over the backward-facing step is considered to be a benchmark problem to study the complicated flow physics such as separation, free shear layers, reattaching flow, recirculation, and high turbulence intensities.

New modification is made in the geometry to study the flow separation. Some early studies were performed [1-5] by modifying the backward-facing step geometry. One of the applications of such sudden expansion geometry can be found inside the combustor [6,7]. The competing challenges of achieving higher turbulent kinetic energy and longer residence time demands modification of the step geometry. A large vortex will provide greater residence time for the fresh reactants to achieve complete combustion. Turbulent kinetic energy improves mixing and better combustion, reduces ignition delay. In this analysis, we have considered non-reacting flow, and modification is made on the geometry in the experiment conducted by *Driver and Seegmiller* [5].

1.1 Experimental Study

The experimental study of flow over the backward-facing step was performed by *Driver and Seegmiller* [5]. This study was selected for validation use in the present work, because of the extensive quantitative measurements made. The experiment set-up consists of a rectangular inlet duct followed by a 1.27 cm backward-facing step on the floor. The height of the inlet duct is 8H and the height of the duct after the step is 9H, where H is the step height. The freestream velocity was 44.2 m/sec (Mach number = 0.128) in standard atmosphere. This test configuration has a small expansion ratio (9H / 8H = 1.125) to minimize the freestream pressure gradient owing to sudden expansion. The boundary layer thickness measured at the location 4H upstream of the step was 1.9 cm. A high Reynolds number will ensure that the boundary layer would be fully turbulent before passing over the step.

2. Project Description

In the first stage, numerical simulation is performed on the 2-D BFS geometry to reproduce the experimental set-up of *Driver and Seegmiller* [5]. According to *Mustafa Kemal Isman* [8], the inlet flow domain is extended upstream by 36H on flow separation. The inlet duct length of 40H will make sure that the boundary layer thickness of 1.9 cm at a distance 4H upstream of the step in the experiment is matched with the CFD simulation to match the experimental condition. The channel length downstream of the step is at 30H in the experiment, whereas in this simulation the outlet is at a location of 60H from the step in the downstream to make sure zero-normal gradient boundary condition is satisfied. Figure 2.1 (*Fig 2.1*) shows the traditional BFS geometry used in the simulation. The kinematic viscosity of the fluid is equal to 1.51 x 10^{-5} m^2/s. The inlet boundary condition has a velocity of 44.2 m/sec. The walls of the simulation test configuration are at no-slip boundary condition. The operating condition is set to 101325 pascals. The simulation results obtained such as reattachment length, skin friction coefficient, velocity profile at different locations, and coefficient of pressure are compared with the experimental data.

The second stage includes the modification of traditional BFS geometry. After reflecting the experimental results in CFD simulation, the modified geometries (*Fig 2.2*) were tested for $\alpha = 0^0$, 25^0, and 45^0 using the previous simulation test condition.

2.1 Numerical Procedure

The numerical simulation is performed on the ANSYS Fluent (software) using Reynolds Averaged Navier-Stokes (RANS) approach. RANS is obtained from averaging the Navier-Stokes equations over a time period ΔT. ANSYS Fluent uses the finite volume method to discretize the partial differential equations. Several turbulence-model were tested and validated by *Kim et al.* [9] for comparison of near-wall treatment methods for flow over a backward-facing step. According to their study, the SST k- ω model was found to be over predict the reattachment length. The RNG k-ε model computed a closer value to the actual experiment than the SST k- ω model. However, in this analysis, the SST k- ω turbulent model is chosen. To compute the separated flow problem, we use a low Reynolds number turbulence model such as Shear Stress Transport (SST) k-ω. This includes two additional transport equations to represent the turbulent properties of the flow. The SST k- ω model accounts for its good behavior in adverse pressure gradient and separating flow. We have used SIMPLE (Semi-implicit method for pressure-linked equations) algorithm for pressure velocity coupling since the flow was considered to be a steady-state. For pressure interpolation, second-order scheme is used to compute the face values of pressure from the cell values. Momentum equations, turbulent kinetic energy, and specific dissipation rate equation are discretized by a second-order upwind scheme which will add accuracy in the solution. The least-square cell-based method is applied for gradient discretization. In this method, the cell gradient is determined by solving the minimization problem for the system of the non-square coefficient matrix in a least-squares sense. The least-squares method does not use the values on the faces of the cell to calculate the gradient, it only uses the distance vectors between the centroid and the centroids of the neighboring cells [10].

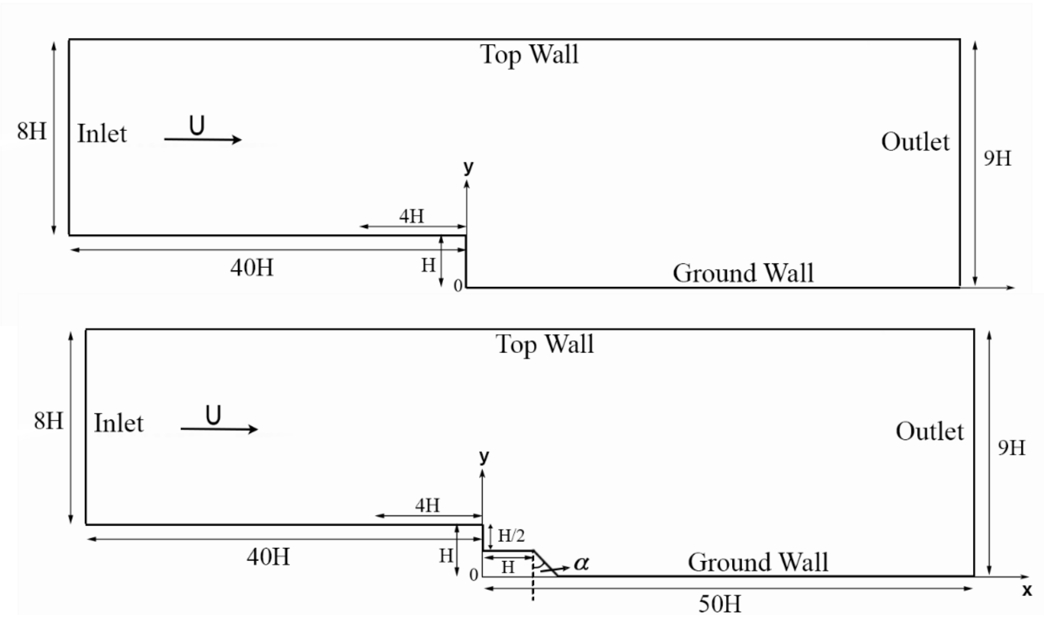

Fig -2.2: Modified backward-facing step (BFS) geometry (step angle $\alpha = 0^0, 25^0, 45^0$)

2.2 Meshing

The near-wall region consists of three sub-regions, namely the viscous sub-layer, the buffer layer, and the fully turbulent region. In the viscous sub-layer, viscosity plays a dominant role in momentum and heat/mass transfer. In this region, flow is almost laminar. While using a low Reynolds number turbulence model to solve the near-wall region, the first cell should be placed in the viscous sub-layer. In turbulence modeling, it is important to determine the proper size of the cells in the near-wall domain. The turbulence model wall laws have restrictions on the y^+ near the wall. y^+ is a non-dimensional wall distance for a wall-bounded flow. It can be defined in the following way-

$$y^+ = \frac{y\mu_\tau}{\vartheta}$$

ϑ is the kinematic viscosity y^+
y is the absolute distance from the wall
μ_τ is friction velocity at the nearest wall

The SST k-ω turbulent model requires the y^+value to be less than 1 to capture the viscous sub-layer. Based on the $y^+ \approx 1$ the y value for the first cell distance from the wall was calculated to be 8.275x10^{-6} m. The blocking method was implemented in ANSYS ICEM CFD (software) to generate the fully structured non-uniform quadrilateral cells. To ensure adequate meshing to resolve the gradients, a sub-layer growth rate of 1.15 was applied. A finer grid is used in the region where the flow behavior is expected to be more complex, such as the expansion zone and recirculation regions. The mesh independence study was performed on traditional backward-facing step geometry to eliminate any possibility of the result depending on the mesh. The initial mesh for traditional geometry (*fig 2.3*) has 42275 quadrilateral cells and the refined mesh has 64465 quadrilateral cells. A similar meshing strategy is applied to modified backward-facing step geometries. *Fig 2.3* and *fig 2.4* is not a complete image of the domain.

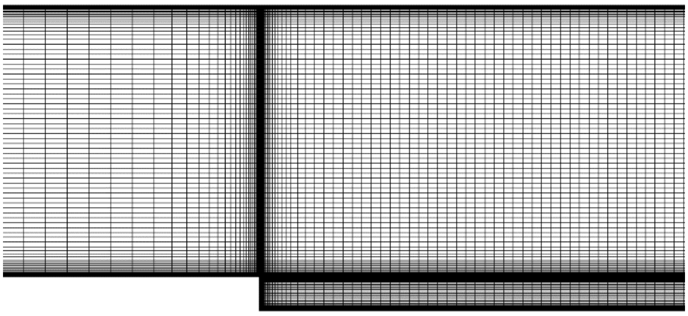

Fig -2.3: Discretization of traditional step geometry (initial mesh)

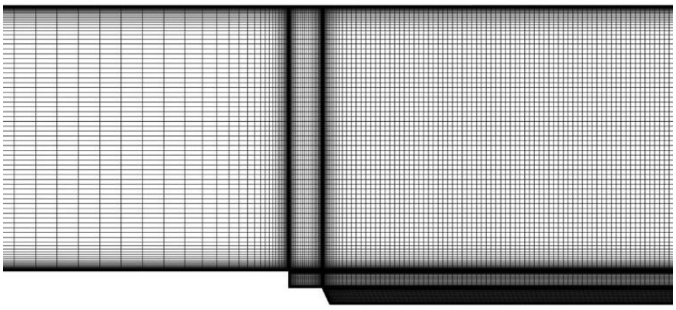

Fig -2.4: Discretization of modified step geometry ($\alpha = 25^0$)

3. Results

All the results presented here are after achieving a converged solution.

1. *Validation of simulation data with experimental data for traditional BFS geometry:*

The boundary layer thickness obtained at a distance 4H upstream from the step is 1.91 cm which matches the experimental value 1.9 cm. The y^+value for the inlet ground wall calculated by the solver is approx. 0.7 which is less than y^+= 1, hence our first cell lies in the viscous sub-layer. Reattachment length is a sensitive parameter often used to quantify the accuracy of the solution to BFS flow. The reattachment point is the location downstream of the step where the flow separates at the step corner and ….

…. reattaches with the bottom ground wall. At this location, wall shear stress is zero and this distance from the point of separation is termed as reattachment length. The experimental value of reattachment length calculated by *Driver and Seegmiller* [5] is x/H = 6.25 while the data obtained from the simulation show x/H = 6.32. The simulation shows a higher value of reattachment length indicating that the result has an error of approx. 1.12% which is less than 5% for validation purpose. The mesh independence test performed showed no major difference in the reattachment length as shown in *Table 3.1*.

Table -3.1: Reattachment length for initial mesh

Traditional geometry	Number of quadrilateral cells	Reattachme -nt length (x/H)	% Error between experimental and CFD results
Experiment	-	6.25	-
Simulation (Initial Mesh)	42275	6.32	1.12 %
Simulation (Refined Mesh)	64465	6.31	0.96 %

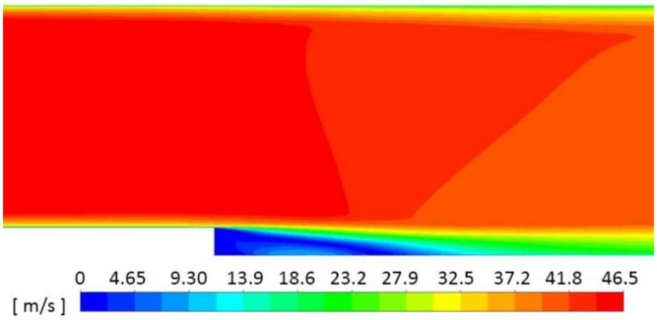

Fig -3.1: Velocity contours

The skin friction coefficient is the ratio of local shear stress to the characteristic dynamic pressure. Since simulation over predicted the reattachment length, the skin friction coefficient (C_f) for the ground wall shows a higher value than the experimental data as shown in *chart 3.1*. The skin friction coefficient (C_f) is negative in the recirculation region and zero at the reattachment point. A pressure coefficient (C_p)is a dimensionless number that describes the relative pressures throughout a flow field. The pressure coefficient on the ground wall has a negative value in the recirculation region showing a decrease in the static pressure after flow separation (*chart 3.2*). The axial velocity profile was predicted reasonably well at the location x/H = 1 downstream of the step (*chart 3.3*).

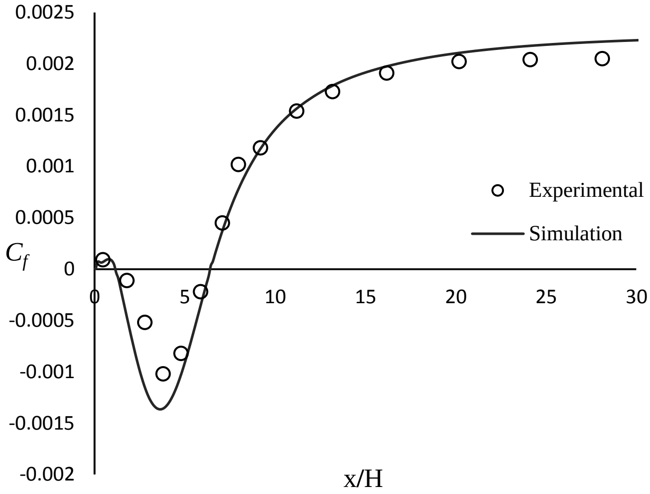

Chart -3.1: Skin friction coefficient distribution

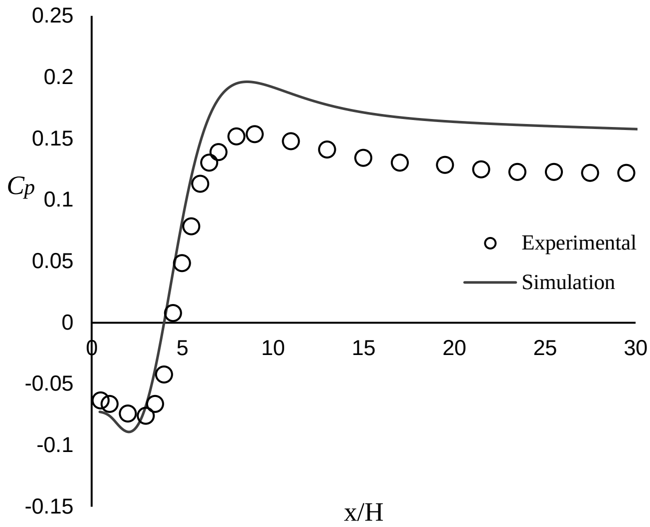

Chart -3.2: Wall pressure coefficient distribution

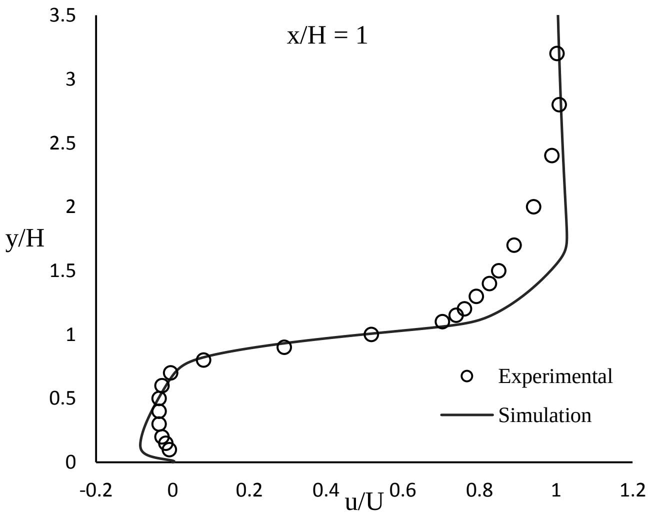

Chart -3.3: Mean velocity profile at x/H = 1

2*. Simulation results for modified BFS geometries:*

The reattachment length for modified BFS geometry for $\alpha = 0^0$, 25^0, and 45^0 are shown in *Table 3.2.* The reattachment length is decreased in the modified geometries as compared to the traditional geometry. This decrease in the reattachment length can be a result of a decrease in adverse pressure gradient due to the division of step. The effect of the expansion ratio on separating and reattaching shear flows was reviewed by *Eaton and Johnston* [11]. As per the study done by *Eaton and Johnston* [11], they observed a trend that the reattachment length increased with the increase in expansion ratio. The division of a single step into two steps has reduced the effect of sudden expansion on the flow.

Table -3.2: Reattachment length in modified geometries

Modified Geometry - Step Angle α	Reattachment Length (x/H)
0^0	6.17
25^0	6.26
45^0	6.17

The turbulent kinetic energy profile at different locations downstream of the step is plotted for all the geometries. Turbulence kinetic energy is the mean kinetic energy per unit mass associated with eddies in a turbulent flow. The turbulent kinetic energy is found to be an increase in the downstream of the step but there was no major effect seen was the same for all the geometries. The maximum TKE value from the simulation at the location x/H = 1, 2, and 3 is mentioned in the *Table. 3.3* for all the geometries tested.

Table -3.3: Turbulent kinetic energy at a different location in the downstream of the step

Geometry	Max. TKE (m^2/s^2) at a different location in the downstream			% Increase in TKE compared to Traditional Geometry		
	x/H= 1	x/H= 2	x/H = 3	x/H= 1	x/H= 2	x/H = 3
Traditional BFS	40.41	47.07	59.8	-	-	-
Modified $\alpha = 0^0$	49.48	52.53	61.16	22.44 %	11.59 %	2.13 %
Modified $\alpha = 25^0$	48.93	52.67	61.05	21.08 %	11.89 %	1.95 %
Modified $\alpha = 45^0$	47.87	54.13	62.29	18.46 %	14.99 %	4.02 %

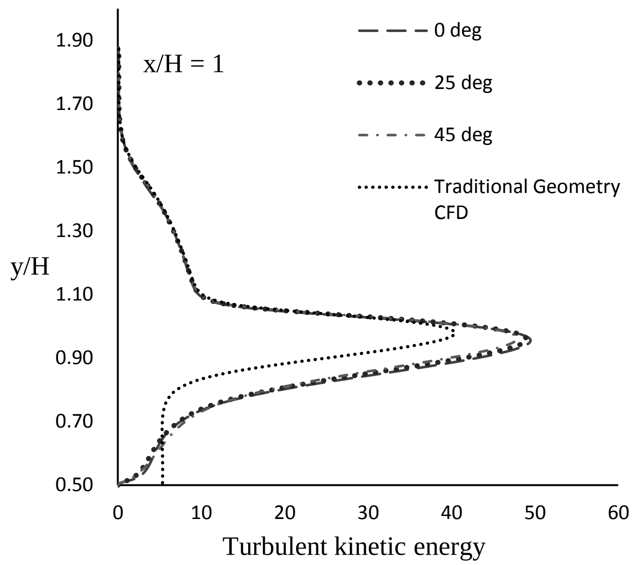

Chart -3.4: Turbulent kinetic energy profile at x/H= 1

In *chart 3.4* the TKE profile at x/H = 1 for traditional geometry shows a lower value compared to the modified geometry TKE profile. The maximum percentage increase is for modified geometry, $\alpha = 0^0$.

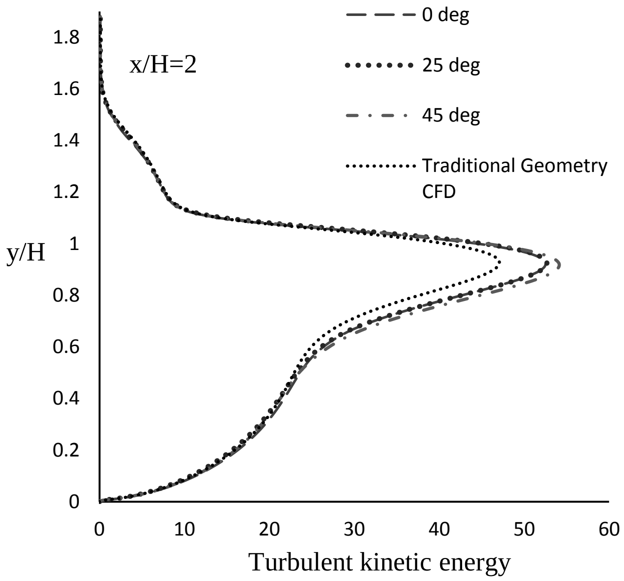

Chart -3.5: Turbulent kinetic energy profile at x/H= 2

In *chart 3.5*, the maximum percentage increase in TKE compared to traditional geometry is 14.99% for modified geometry, $\alpha = 45^0$ geometry. The lowest value of TKE is for traditional geometry. The percentage increase for $\alpha = 0^0$ and 25^0 is around 11%.

5. Conclusion

The numerical analysis of the flow over a modified backward-facing step leads to the following conclusions:

1) The reattachment length for modified geometry is x/H = 6.17, 6.26, and 6.17 for $\alpha = 0^0$, 25^0, and 45^0. This decrease in reattachment length has a negative effect, i.e. it reduces the recirculation region which leads to a decrease in the residence time of fresh reactant.

2) A favorable effect was observed on the turbulent kinetic energy. The division of step resulted in an increased turbulent kinetic energy in the downstream region of the step. Simulation results shows an increase in turbulent kinetic energy of 22%, 21%, and 18% for $\alpha = 0^0$, 25^0, and 45^0 respectively at a location x/H = 1. This increase in turbulent kinetic energy will improve the mixing and burning velocity of fresh reactants. Further downstream at x/H = 3 the increase in turbulent kinetic energy is 2%, 1.9%, and 4% for $\alpha = 0^0$, 25^0, and 45^0 respectively. Beyond x/H = 3, there is a very slight or no increase in turbulent kinetic energy.

3) Hence, there is a contradictory effect on the division of step i.e. decrease in reattachment and increase in turbulent kinetic energy. But the decrease in reattachment length is around 0.94%-2.37% for modified geometries

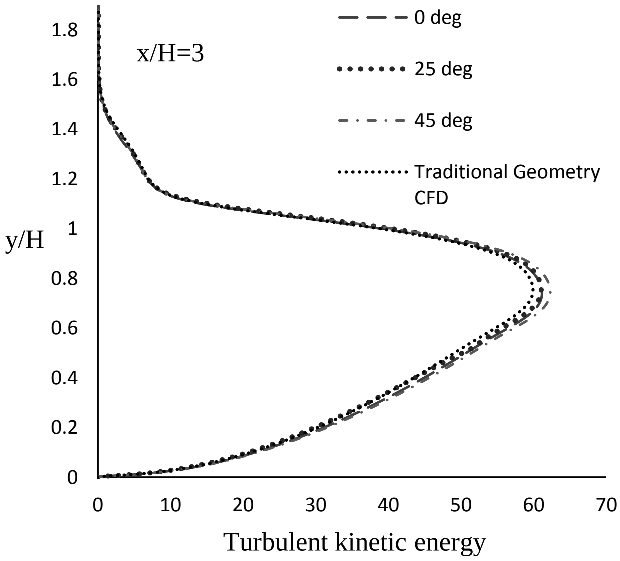

Chart -3.6: Turbulent kinetic energy profile at x/H= 3

Chart 3.6 shows the TKE profile at x/H = 3, at this location, there is no major difference in TKE profile for traditional geometry and modified geometries. Beyond this location downstream there are no changes observed in TKE.

And the maximum increase in turbulent kinetic energy is around 18%-22% (at x/H = 1), therefore more favorable results have been observed on the backward-facing step modified geometry.

The conclusion can be given on the division of step but not on the changes in the step angle (α) of modified geometry. Since it does show any strict variation. In this analysis, we considered a non-reacting flow. The dynamics of the flow should also be tested in the presence of reacting flow for further conclusions.

REFERENCES

1. Choi, H.H. and Nguyen, J., 2016. Numerical investigation of backward facing step flow over various step angles. Procedia Engineering, 154, pp.420-425.
2. Wang, S., Burton, D., Sheridan, J. and Thompson, M.C., 2014, December. Characteristics of Flow over a Double Backward-Facing Step. In 19th Australasian Fluid Mechanics Conference.
3. Nice, G.R., Tseng, W.Y. and Moses, H.L., 1966. Separation of turbulent, incompressible flow from a curved, backward-facing step. Cambridge, Mass.: Massachusetts Institute of Technology, Gas Turbine Laboratory, [1966].
4. Tinney, C.E. and Ukeiley, L.S., 2009. A study of a 3-D double backward-facing step. Experiments in fluids, 47(3), pp.427-438.
5. Driver, D.M. and Seegmiller, H.L., 1985. Features of a reattaching turbulent shear layer in divergent channel flow. AIAA journal, 23(2), pp.163-171.

6. Sengupta, K., Russell, K. and Mashayek, F., 2007. Step geometry and counter current effects in dump combustor, Part 1: Cold flow. AIAA journal, 45(8), pp.2033-2041.
7. So, R.M. and Ahmed, S.A., 1989. Characteristics of dump combustor flows. International journal of heat and fluid flow, 10(1), pp.66-74.
8. Isman, M.K., 2016. Investigation of inlet effects on backward-facing step flow prediction. Transactions of the Canadian Society for Mechanical Engineering, 40(3), pp.317-329.
9. Kim, J.Y., Ghajar, A.J., Tang, C. and Foutch, G.L., 2005. Comparison of near-wall treatment methods for high Reynolds number backward-facing step flow. International Journal of Computational Fluid Dynamics, 19(7), pp.493-500.
10. Syrakos, A., Varchanis, S., Dimakopoulos, Y., Goulas, A. and Tsamopoulos, J., 2017. A critical analysis of some popular methods for the discretisation of the gradient operator in finite volume methods. Physics of Fluids, 29(12), p.127103.
11. Eaton, J.K. and Johnston, J.P., 1981. A review of research on subsonic turbulent flow reattachment. AIAA journal, 19(9), pp.1093-1100.

NOMENCLATURE

$\mathbf{C_f}$ Skin Friction Coefficient ($\boldsymbol{\tau_w}/0.5\boldsymbol{\rho}\mathbf{U^2}$)

$\mathbf{C_p}$ Coefficient of Pressure $(\mathbf{P}-\mathbf{P_0})/0.5\boldsymbol{\rho}\mathbf{U^2}$

H Step Height

k Turbulent Kinetic Energy

P Static Pressure

$\mathbf{P_0}$ Reference Pressure (x/H= -4)

U Free Stream Velocity (44.2 m/sec, Mach- 0.128)

u Velocity in x-Direction

$\mathbf{y^+}$ Non-Dimensional Wall Distance

α Step Angle (0^0, 25^0, 45^0)

ρ Density (1 kg/m^3)

ϑ Molecular Viscosity (1.5 x10^{-5} m^2/s)

$\boldsymbol{\tau_w}$ Shear Stress

ω Specific Turbulent Dissipation Rate

BFS Backward-facing Step

SST Shear Stress Transport

TKE Turbulent Kinetic Energy