Effect of Streamline Curvature on Heat Transfer in Turbulent Flow over Backward Facing Step

P Sajesh¹ and J S Jayakumar²
¹²Department of Mechanical engineering, Amrita Vishwa Vidyapeetham, Amritapuri, India
E-mail: ¹sajeshp92@gmail.com and ²jsjayan@gmail.com

Abstract. The present investigation is to find the enhancement of heat transfer in the recirculation region when additional centrifugal forces due to streamline curvature are imposed on to the shear-layer. Backward Faced Step (BFS) have got important importance to study the separated and recirculating flows and their heat and mass transfer effects due to its geometrical simplicity. The effects of the radial pressure gradient existing in case of the curved stream line and how it affects the turbulence are investigated in this present study. The stabilizing and destabilizing effects with heat transfer have been studied in this paper for three different curvatures (flat, concave, and convex). Numerical results obtained by the use of the open source code OpenFOAM-3.0.1 is compared with the experimental results. The three-dimensional analysis has been done using an aspect ratio of 18 (AR=18:1) at Reynolds number based on step height as 80000. The distribution of Nusselt number (Nu), skin friction coefficient (Cf), and coefficient of pressure (Cp) were found on the bottom wall with close heed. The results show that the convex surface stabilizes the flow whereas the concave surface introduce more disturbances and destabilizes the flow.

1. Introduction

Since the last decade, backward faced step has got great importance in the numerical study of flow separation in internal flow due to sudden geometric change. Flow separation and reattachment occur in many applications. Some of them are in power engineering equipment, flow over airfoils at large angles of attack, diffusers, air flow over fences and in pipes whose area suddenly increases. Flow area simplicity of the backward faced step is the major reason for selecting this configuration as representative geometry. Another main feature of the BFS is that the point of separation is a fixed one. The basic flow physics of flow separation and recirculation can be easily visualized and studied with this basic geometry. The importance of such flow in engineering stream is stressed in many works (see e.g. Iwai et al. [6], Goldstein et al. [9], and Chen et al. [10]), and attempts have been made to develop advanced experimental and theoretical techniques in order to study the flows with separation regions. Literature shows that lot of flow modifications has been studied in the field of flow over BFS, such as inclined step, multiple step and vertical step.

There exists two types of curvature problems in case of flow through curved channels. They are classified as longitudinal curvature problem (swirling flows) and transverse curvature problem (Flow in curved ducts). The appearance of streamline curvature can change the turbulent flow structures severely. The change in flow structure will influence the mean flow field and vice versa. Bradshaw et al. [1] found out that the convex curvature reduces the turbulence intensity which stabilizes the flow, while the concave curvature introduce more longitudinal vortices which in turn increases turbulent intensity and
turbulent mixing that assists the destabilizing effect. In particular, most of the common problem in BFS with various flow situations have been analysed experimentally and numerically to obtain the basic information of the flow separation and reattachment phenomena. All existing work related to this topic are experimental and used to evaluate the effect of streamline curvature on flow stability not on the reattachment length and recirculation. But this work is done to evaluate the effect of streamline curvature on heat transfer and flow stability in BFS using numerical simulations.

2. Governing Equations

Finite volume method is used to numerically solve the time averaged Navier Stokes and energy equations along with the continuity equations using k-ω SST model, since the model exhibits good behavior in adverse pressure gradient flow situations.

Continuity Equations

\[
\frac{\partial \rho}{\partial t} + \frac{\partial \rho u}{\partial x} + \frac{\partial \rho v}{\partial y} + \frac{\partial \rho w}{\partial z} = 0
\]  

(1)

x-Momentum Equation

\[
\rho \frac{\partial u}{\partial t} + \rho u \frac{\partial u}{\partial x} + \rho v \frac{\partial u}{\partial y} + \rho w \frac{\partial u}{\partial z} = \rho g_x - \frac{\partial p}{\partial x} + S_{dr} + S_w + \frac{\partial}{\partial x} \left[ 2 \mu \frac{\partial u}{\partial x} - \rho u u \right] + \frac{\partial}{\partial y} \left[ \mu \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) - \rho u v \right] + \frac{\partial}{\partial z} \left[ \mu \left( \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right) - \rho u w \right]
\]  

(2)

y-Momentum Equation

\[
\rho \frac{\partial v}{\partial t} + \rho u \frac{\partial v}{\partial x} + \rho v \frac{\partial v}{\partial y} + \rho w \frac{\partial v}{\partial z} = \rho g_y - \frac{\partial p}{\partial y} + S_{dy} + S_w + \frac{\partial}{\partial x} \left[ \mu \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) - \rho u v \right] + \frac{\partial}{\partial y} \left[ 2 \mu \frac{\partial v}{\partial y} - \rho v v \right] + \frac{\partial}{\partial z} \left[ \mu \left( \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \right) - \rho v w \right]
\]  

(3)

z-Momentum Equation

\[
\rho \frac{\partial w}{\partial t} + \rho u \frac{\partial w}{\partial x} + \rho v \frac{\partial w}{\partial y} + \rho w \frac{\partial w}{\partial z} = \rho g_z - \frac{\partial p}{\partial z} + S_{dz} + S_w + \frac{\partial}{\partial x} \left[ \mu \left( \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right) - \rho u w \right] + \frac{\partial}{\partial y} \left[ \mu \left( \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \right) - \rho v w \right] + \frac{\partial}{\partial z} \left[ 2 \mu \frac{\partial w}{\partial z} - \rho w w \right]
\]  

(4)

Energy Equations

\[
C_p \frac{\partial T}{\partial t} + \rho C_p U \frac{\partial T}{\partial x} + \rho C_p V \frac{\partial T}{\partial y} + \rho C_p W \frac{\partial T}{\partial z}
\]

\[
= \frac{\partial}{\partial x} \left[ k \left( \frac{\partial T}{\partial x} \right) - \rho C_p u' T' \right] + \frac{\partial}{\partial y} \left[ k \left( \frac{\partial T}{\partial y} \right) - \rho C_p v' T' \right] + \frac{\partial}{\partial z} \left[ k \left( \frac{\partial T}{\partial z} \right) - \rho C_p w' T' \right] + q_v
\]  

(5)

Boussinesq approximation that defines the eddy viscosity and eddy conductivity as follows.

\[
\mu = \frac{-\rho u u}{2 \frac{\partial u}{\partial x}} = \frac{-\rho v v}{\frac{\partial v}{\partial y} + \frac{\partial v}{\partial x}} = \frac{-\rho w w}{\frac{\partial w}{\partial z} + \frac{\partial w}{\partial y}} = \ldots
\]  

(6)

\[
k = \frac{-\rho C_p u u}{\frac{\partial u}{\partial x}} = \frac{-\rho C_p v v}{\frac{\partial v}{\partial y}} = \frac{-\rho C_p w w}{\frac{\partial w}{\partial z}}
\]  

(7)
3. Methodology
The flow field corresponding to the simulation is shown in the figure 1. The three dimensional geometry consists of 1500 mm straight development section and 2000 mm curved test section. Concave and convex curvatures are provided for the curved test section with various radiuses of curvatures which is shown in the Table.1. For both the curvatures (weak and mild curvatures) the test section consists of a duct aspect ratio (AR) of 18 and an expansion ratio (ER) of 1.5. Three type of test sections were numerically simulated for a turbulent Reynolds number of 80000 based on the step height.

![Figure 1. Computational Flow Domain - Honami et al.[8].](image)

Study of the effect of curvature on the enhancement of heat transfer on bottom wall of downstream test section is carried out by providing uniform temperature on it and the analysis is treated as a constant wall temperature problem. Prandtl number of 0.7 is considered for a film temperature of 53˚C, which is used to calculate the flow properties. Air is taken as the working medium which is treated as incompressible under these conditions. To study the heat transfer phenomenon a new solver is developed, known as tempisoFoam, which is formed from existing pisoFOAM solver by incorporating Energy equations.

Table 1. Test Section Curvatures.

| R(mm) | Flat | Convex | Concave |
|-------|------|--------|---------|
|       | Weak | Mild   | Weak    | Mild   |
| ∞     | 1600 | 800    | 1720    | 920    |

4. Boundary Conditions
The boundaries of the flow domain are divided as inlet, outlet, step, bottom, top, and frontAndBack. The inlet velocity is specified as 36.48 m/s to have a turbulent Reynolds number of 80000. The rest of the boundaries are considered to be having a No slip boundary condition except outlet. The turbulence data is provided based on the following equations from (8) to (9), where the main free stream is in the order of 0.2%.

Turbulent Kinetic Energy: \[ k = 1.5 \times (U \times I)^2 \] (8)

Turbulent Dissipation Rate: \[ \varepsilon = 0.094 \times \frac{3}{0.07} \times \frac{k^2}{L_c} \] (9)

Specific Dissipation Rate: \[ \omega = \frac{\varepsilon}{k} \] (10)
5. Validation

Grid independent study was done by calculating the reattachment length for a flat plate situations for various number of cells. The measured reattachment length in the backward facing step containing flat test section become independent of the grid size after the number of cells become 3000000. This flow domain is adapted for the numerical simulation of the other curvatures.

The tempisoFoam solver is selected for the numerical simulation of the curved test section and k-ω SST model is adapted to solve turbulence flow problem. Figure 2 shows the numerical results of wall static pressure distribution on the center line of the bottom wall, which is validated with experimental results given by S.Honami et al [8]. Once the tempisoFoam solver is validated with the experimental results, the same solver is used to evaluate the effect of stream line curvature on heat transfer and flow stability for various curvatures.

![Figure 2. Validation.](image)

6. Results and Discussions

In a curved streamline, the pressure in the outer streamline will be higher than the inner one. Due to pressure gradient present across the curved streamlines, the flow will follow a curved wall as possible (if the curvature is not sufficient the flow get separated and forms recirculation region as in backward faced step). Figure 3 shows the static pressure distribution for different surface curvatures in the primary recirculation region. Results of the $C_{px}$ values on the bottom wall shows that the convex curvature with smaller radius of curvature ($R$) exhibits lowest pressure distribution and concave curvature with smaller $R$ exhibits the highest pressure distribution. The cause of this static pressure distribution is due to the different pressure gradient existing across the curved streamlines at a particular cross section.

![Figure 3. Wall static pressure distribution for different curvatures.](image)
The flow direction gets changed as the skin friction coefficient changes its sign from positive to negative. Initial sign change in figure 4 describes the presence of corner eddies near the step, while the further change in $C_f$ downstream of the step indicates the reattachment point. Pressure gradient existing in the convex curvature is pushing the flow towards the bottom wall whereas in concave curvature it tries to pull the flow in the test section. This is the reason why the $C_f$ values of convex curvature shifts towards the left and concave curvature shifts towards right compared to the flat configuration.

**Figure 4.** $C_f$ Distribution along centerline for different curvatures.

The length of the corner eddies are not significantly affected by the surface curvatures compared its effect on reattachment length. The effect of surface curvature on the corner eddies is shown in Table 2. Length of the corner eddies become increased when the concave curvature is imposed on the test section and convex curvature suppresses them.

| Curvature       | Length of corner eddies: $X_c/S$ |
|-----------------|----------------------------------|
| Convex Mild     | 1.15                              |
| Convex Weak     | 1.23                              |
| Flat            | 1.29                              |
| Concave Mild    | 1.34                              |
| Concave Weak    | 1.365                             |

The numerical results captures the primary recirculation region formed downstream of the step due to the flow separation. The dependence of surface curvature on the reattachment length is discussed in Table 3. The separated flow gets attached to the downstream of the test section and reattachment length affected in the same fashion as the curvature affects the corner eddies. Primary recirculation region become extended downstream due to pressure gradient and destabilizing effect of the concave curvature.

| Curvature       | Reattachment length: $X_r/S$ |
|-----------------|------------------------------|
| Convex Mild     | 6.39                         |
| Convex Weak     | 6.62                         |
| Flat            | 6.77                         |
| Concave Mild    | 6.98                         |
| Concave Weak    | 7.01                         |

The $C_f$ contours downstream of the step is shown in the figure 5 for different curvature of the test section. The stream wise skin friction coefficient lines will be used to characterize the flow direction of the fluid layer adjacent to the test section. The red lines in the diagram, where the skin friction coefficient is positive, represents the primary recirculation region. The primary recirculation region is characterized
by the fluid flow towards the step, hence wall shear stress is acting along the flow direction. This is the reason for the skin friction coefficient to have a positive value in the recirculation region. S. Honami et al [8] experimentally evaluated the effect of streamline curvature on reattachment length. The numerical results obtained by this study is in good agreement with the experimental results of S. Honami et al. Inherent stability of the straight channel, stabilizing and destabilizing effects of the convex and concave curvature on the reattachment process is clearly visible from the $C_f$ contours. $C_f$ contours exhibits the disturbances in the flow regime, when the additional centrifugal force due to streamline curvature is imposed on the shear layer. Wavy nature of the $C_f$ contours in case of the concave curvature is due to the higher destabilizing effect, which resembles the theory explained by Bradshaw et al [10] based on the Richardson Number ($Ri$),

$$Ri = \frac{\lambda (\frac{\partial U}{\partial n})}{U_0 n}$$  \hspace{1cm} (8)

Where $R$ is the radius of curvature, $n$ is the normal distance away from the wall, and $U_0$ is the longitudinal velocity. For a concave surface with lower radius of curvature, the Richardson number become higher which aids the apparent mixing length and destabilizes the flow. Hence the $C_f$ contours along the span wise position become drastically fluctuating in case of a concave curvature.

![Figures](a) Concave Mild  
(b) Concave Weak  
(c) Convex Mild  
(d) Convex Weak
Figure 5. Cf contours on the bottom wall for different curvatures.

Figure 6. Nu distribution along centreline.

Nu distribution on the bottom wall downstream of the step is shown in the Figure 6. Maximum Nu always exists downstream of the reattachment point in all five cases. Peak Nu value on the bottom wall indicates the max heat flux from the heated wall. Nu distribution shows that the peak Nu value is obtained for convex curvature due to its stabilizing effect on turbulence. Convex curvature of the streamlines in the boundary layer dampens the turbulence, especially the shear stress and Reynolds stress normal to the wall.

Numerical results of Nu contours on the bottom wall is shown in figure 7. Maximum heat transfer is occurred towards the side wall, not on the center line of the bottom wall, in all the cases studied here. H. Iwai et al [6] also noted the same observation of occurrence of maximum Nu on the bottom wall of a flat BFS at two points which are symmetrically situated near the side walls. The maximum Nu value of 181.9 is obtained for the convex streamline with mild curvature. There exists a span wise pressure gradient, which is symmetrically decreases towards the center line of the duct. Hence the fluid particle from the side walls are recirculate spirally towards the center line in the recirculating zone. This observation closely resembles the experimental results obtained by Goldstein et al [9]. This spirally recirculating flow enhances the heat transfer near the side walls. Enhancement of heat transfer is
observed by the red zone in the recirculation flow region in the Nu contours. Presence of corner eddies suppresses the heat transfer which is shown as the blue color near the step.

Figure 7. Nu contours on the bottom wall for different curvatures.
7. Conclusions
Numerical simulations has been done for the turbulent flow over the BFS with different surface curvatures. The results obtained are concluded as follows,

- There exists a pressure gradient across the curved streamline, which is decreasing towards the center of curvature.
- Turbulence will be stabilized due to the presence of convex curved streamlines in the boundary layer flow and the concave nature of the streamlines in the boundary layer flow will destabilizes the flow.
- The peak Nusselt number on the bottom wall is obtained near to the side wall, but not on the centerline.
- The occurrence of peak Nusselt number towards the side walls are due to the presence of spirally recirculating flow regime present on the downstream of the flow.
- Convex curvature text section exhibits the maximum heat transfer on the centerline of the bottom wall due to its stabilizing effect.

8. References
[1] Muck K C, Hoffmann P H, and Bradshaw P 1985 J. of Fluid Mechanics. The Effect of Convex Surface Curvature on Turbulent Boundary Layers, Vol. 161, pp 347-369.
[2] Hoffmann P H, Muck K C, and Bradshaw P 1985 J. of Fluid Mechanics, The Effect of Concave Surface Curvature on Turbulent Boundary Layers, Vol. 161, pp 371-403.
[3] Bradshaw P and Meroney R N 1975 AIAA Journal, Turbulent Boundary-Layer Growth over a Longitudinal Curved Surface, Vol. 13, pp 1448-1453.
[4] Davidson Lars. Fluid mechanics, Turbulent Flow and Turbulence Modeling, Chalmers University of Technology, Goteborg, Sweden.
[5] Hunt I A and Joubert P N 1979 J. of Fluid Mechanics, Effects of Small Streamline Curvature on Turbulent Duct Flow, Vol. 91, pp 633-659.
[6] Iwai H, Nakabe K and Suzuki K 2000 Int. J. of heat and mass transfer, Flow and heat transfer characteristics of backward-facing step laminar flow in a rectangular duct, 43(3), pp 457-471.
[7] Armaly B, Durst F, Pereira J and Schönung B 1983, J. of Fluid Mechanics, Experimental and theoretical investigation of backward-facing step flow, 127, pp 473-496.
[8] Honami S and Nakajo I 1983, Eighth Australasian Fluid Mechanics Conference, University of Newcastle, N.S.W, The effect of streamline curvature on the reattaching turbulent flow over a backward-facing step.
[9] Goldstein R J, Eriksen V L, Olson R M and Eckert E G 1970, ASME. J. Basic Eng, Laminar Separation, Reattachment, and Transition of the Flow over a Downstream-Facing Step, 92(4), pp 732-739.
[10] Chen Y T, Nie J H, Hsieh H T, and Sun L J 2006, Int. J. of Heat and Mass Transfer, Three-dimensional convection flow adjacent to inclined backward-facing step, 49(25-26), pp 4795-4803.
[11] Santhosh S 2018, ICTEA: International Conference on Thermal Engineering, Effects of Buoyancy on the Reattachment Length in Flow over Heated Vertical Backward Facing Step.