On the separated region behind a confined backward-facing step

J.N. Fernando, J. Kriegseis and D.E. Rival

Abstract

The separated flow behind a three-dimensional backward-facing step (BFS) is examined using numerical and experimental techniques. This is accomplished through Reynolds-Averaged Navier-Stokes (RANS) simulations and time-averaged particle image velocimetry (PIV) measurements. Vortical structures behind the constrained backward-facing step are characterized using each approach and the reattachment line downstream of the step is determined through the numerical simulations. This result is compared to data available in the literature for further validation. Topological differences are found to exist between time-averaged solutions of each approach. However, similar vortical structures are present in both solutions. These structures are only visible in the PIV data through image sequence processing and are found to be highly unsteady. The presence of this obvious unsteadiness in the flow demonstrates the limited validity of implementing time-averaged schemes for the prediction of three-dimensional structures behind the backward-facing step.

1 Introduction

Flow separation resulting from a strong adverse pressure gradient due to a sudden expansion in geometry, such as a backward-facing step, occurs in a large array of engineering applications. Many studies on separated flows over the last few decades have focused on the two-dimensional, backward-facing step geometry, as a means of understanding shear-layer behavior and reattachment properties [1–3]. Although the geometry is simple, the flow immediately downstream of the step contains vortical structures and complicated topology.

Inside the recirculation zone the flow is characterized by flow reversals and vortical structures [2–7]. Specifically, the flow phenomenon responsible for vortical structures in the shear layer results from Kelvin-Helmholtz instabilities and is caused by the interaction between the shear layer and recirculating flow near the step wall [3]. In addition to the Reynolds number, the key variables associated with the backward-facing step problem are the expansion and aspect ratios, defined as the ratios of downstream to upstream channel height and channel width to step height, respectively. In constrained (three-dimensional) channels, sidewall effects produce complex flow structures, and for the case of laminar flow, these structures are found to intensify for increasing relative step height and Reynolds number at the channel lower wall for expansion and aspect ratios of 2 and 20, respectively [4, 5].

In the present study, the investigations of the sidewall effects are to be studied in a turbulent flow. The major objective of the current study is to demonstrate how vorticity and coherent structures develop in the flow through both numerical and experimental techniques. Furthermore, the limitations of time-averaged approaches for the backward-facing step problem will be discussed.

2 Geometry

The geometry used in this study is shown in Figure 1. The channel dimensions are \( h = 0.315 \text{ m} \), \( H = 0.45 \text{ m} \), \( s = 0.135 \text{ m} \), \( L_R = 1.78 \text{ m} \), \( L_S = 2.22 \text{ m} \) and \( c = 0.387 \text{ m} \). The inflow and outflow channel heights are given by \( h \) and \( H \), respectively. \( L_R \) and \( L_S \) are corresponding step and downstream channel lengths, \( s \) is the step height and \( c \) is the channel width. The expan-
Fig. 1: Schematic of test geometry, containing a description of relevant dimensions of the confined, backward-facing step.

The expansion ratio is defined as

\[ ER = \frac{H}{h} \]  

and is equal to 1.4. Similarly, the aspect ratio, defined as

\[ AR = \frac{c}{s}, \]  

is equal to 2.67. The origin of the coordinate system is placed at the left-hand corner of the backward-facing step base (Figure 2).

Simulations were performed at a step height of 0.135 m. The Reynolds number, based on the step height \( s \) was evaluated to be approximately 60,000. Velocity at the inlet of the contraction (see Figure 3) was set at 0.05 m/s, with an eddy length scale of 0.45 m, and turbulence intensity of \( Tu = 1\% \). Two other simulations were also performed, varying both turbulence intensity from 1\% to 10\% and velocity from 0.05 m/s to 0.025 m/s so as to determine their contribution (if any) to the flow. These settings were applied as a boundary condition at the inlet cross section, as shown in Figure 3. In this sensitivity study, it was found that although slight changes were observed in magnitude contours of velocity and vorticity - the two variables of greatest interest - neither the turbulent intensity nor the free-stream velocity changed the overall flow topology.

### 3 Numerical Set-up

The complete water-tunnel plenum and channel geometry were modeled in ICEM CFD and then imported into ANSYS CFX. After mesh refinement along the channel boundaries, so as to resolve the viscous sub-layer, the number of nodes in the model totaled 2,197,500. The Reynolds-averaged Navier-Stokes (RANS) equations were solved together with the \( k-\varepsilon \) turbulence model using the ANSYS CFX commercial package. Figure 2 shows a close-up view of the mesh refinement in the region of interest.

### 4 Experimental Method

The experiments were performed on a closed-loop water tunnel, as shown in Figure 4. Water was pumped to a main plenum chamber and through four conditioning units (one honeycomb and three fine screens), before being accelerated into the test section through a six-to-one ratio contraction. The channel ended with a reservoir, which facilitated the recirculation of water back to the plenum through the pump.

To validate the results from the numerical simulations, PIV experiments were performed in the XZ-plane near the step geometry. The experimental set-up consisted of a New Wave Nd:YAG laser (\( \lambda = 532 \) nm) and PCO CCD Camera (1280 x 1024 pixels, 4fps) equipped with a Nikon lens (\( f = 20\text{mm}/2.8 \)). For the data set presented in this paper, 2500 image pairs (\( \Delta t = 24 \) ms) were post-processed using the standard LaVision multi-pass cross-correlation scheme.
Figure 4: Schematic of water tunnel used for PIV experiments. The pump, located at the base of the plenum, drives water through the contraction into the test sections and recirculation chamber.

The set-up can be seen in Figure 5, which depicts PIV data acquisition for a slice in the XZ-plane. Prior to data acquisition, the PIV system was used to carefully adjust the mean tunnel speed with that from the CFD so as to match the operating Reynolds number. Similarly, turbulence intensity of the mean flow in the test section near the step was measured. It was found that the turbulence intensity was approximately 3%. At the same position in the CFD simulation, the average turbulence intensity had an average value of 5%. The sensitivity analysis that was performed as part of the numerical study suggested that this inconsistency would not result in any topological discrepancies between the numerical and experimental simulations.

Figure 5: Detailed sketch of the step including the global origin located at the bottom-left corner of the step; PIV setup and field of view are added for the sake of clarity.

5 RESULTS AND DISCUSSION

One of the main flow features resulting from separated flow off a confined backward-facing step is the primary recirculation reattachment line. This line has been quantified both numerically and experimentally in previously by Nie and Armaly [6], and serves as the main verification measure of the present study. Results for the reattachment line from Nie and Armaly [6] were plotted against the findings from numerical simulations in this study. Figure 6 shows these data sets compared with one another.

Figure 6: Reattachment line for three expansion ratios in the laminar regime (Nie and Armaly [6]), and present work evaluated for an expansion ratio of ER = 1.4 in the turbulent regime.

Although there are clear differences between the reattachment line contours from each study, at least the predicted length of the separated region is in good agreement. The flow topology for the confined backward-facing step in this study suggests a much more unsteady and vorticity-driven flow, as justified by the large differences in the operating Reynolds numbers between the two studies. Although the precise flow topology for this problem has not yet been fully-quantified, a better understanding of the presence of vorticity within the flow will play an important role in understanding the physics of the reattachment length. Accurately determining the reattachment line for this problem may warrant mesh-refinement in the streamwise direction to reduce excessive numerical diffusion. It is also expected that replacement of the two-equation models with more accurate Reynolds-stress models in the numerical solver will improve accuracy of the (spanwise) reattachment length distribution. However, these efforts are beyond the scope of the present study.

Two sets of PIV measurements were taken using the tunnel centerline as the dividing line between data sets, which provided the most advantageous field-of-view (FOV) for direct comparison with the CFD. PIV data was taken in the XZ-plane at the location y/s = 0.5. A comparison of velocity contours in this region is shown in Figure 7.

Velocity magnitude is of the same order but with significant variation in distribution. Topological differ-
ferences were found to exist between time-averaged solutions for both the simulations and experiments. In the numerical solution, two counter-rotating vortices, symmetric about the center line, are evident in the flow. However, for the PIV only streamline curvature is noted, again symmetric about the centerline.

This discrepancy in the flow pattern between PIV and CFD warrants a discussion into the vorticity levels present in this region of the flow. Figure 8 shows a comparison between PIV and CFD results using streamlines and vorticity contours instead.

Although the region of greatest vorticity was almost identical in both PIV and CFD, precise topological flow features (as indicated by velocity vectors), and vorticity magnitude, were found to differ to a significant extent. Furthermore, topological dissimilarity was noted between the time-averaged solutions in both the simulations and experiments. Flow structure variation is evident in Figure 8 through the location of topological singularities (see Foss [8]) in the flow, such as saddles (S) and nodes (N) as well as through the regions of peak vorticity.

In the numerical simulations, the large-scale back-flow forms a vortex in the XZ-plane. When the recirculating flow hits the step, a pair of symmetric nodes (counter-rotating vortices) is generated, and movement of the fluid in the vertical (y-direction) only occurs at the node (N). In contrast, the flow topology of the PIV data suggests that the back-flow after hitting the step is redirected towards the center plane, which leads to flow in the vertical direction in proximity of the center plane.

Time-averaging the results is responsible for the removal of salient features in the flow, causing unsteady and coherent flow patterns to vanish. At certain time intervals in the data set, the formation and subsequent diffusion of vortical structures is visible, revealing similar velocity and vorticity levels as found in the CFD results. This observation is demonstrated in Figure 9 where a sequence of four PIV snap-shots is shown.

Vortical structures analogous to the CFD results are visible in the PIV data, although such occurrences are highly unsteady (Figure 9). The presence of such vortices in the flow, which share order of magnitude agreement with numerical results, demonstrates the contribution of unsteady flow features to the overall velocity average. This insight furthermore emphasizes the limited validity of implementing (time-averaged) RANS simulations for the prediction of three-dimensional structures in the region behind the backward-facing step.

6 Conclusions

We have investigated the separated region behind a confined backward-facing step geometry for a Reynolds number of approximately 60,000 and constant expansion and aspect ratios of 1.4 and 2.67, respectively. The purpose of this study was to gain an understanding into the flow topology of the confined backward-facing step geometry using steady-state RANS schemes. This was accomplished through the use of the standard $k-\varepsilon$ turbulence model and time-averaged PIV data collection. It is apparent from this study, however, that the flow is subject to unsteadiness, especially beneath the shear layer where the flow regime is not easily determined. The direct comparison of the time-averaged results (Reynolds decomposition, see Adrien et al. [9]) of PIV and CFD showed...
significant differences of the flow topology. Consequent analysis of the formation and diffusion of vortical structures in the flow through PIV image sequence processing demonstrated the presence of large-scale coherent flow patterns. This observation retrospectively implies that time-averaging the flow information (for both approaches) is in fact a strong oversimplification, at least with respect to the flow features within the recirculation zone. Therefore, upcoming analysis will involve the application of URANS simulations alongside time-resolved PIV experiments. Further post-processing by means of proper orthogonal decomposition will provide insight into the presence and contribution of the unsteady coherent patterns superimposed on the averaged flow field [10].

ACKNOWLEDGEMENTS

The authors gratefully acknowledge financial support from Canadian Projects Limited (CPL) and the Natural Sciences and Engineering Research Council (NSERC) for this project.

REFERENCES


