A flow analysis for a turning rapid diffuser using CFD

Cuong K. Nguyen\textsuperscript{a}, Tuan D. Ngo\textsuperscript{a}, Priyan A. Mendis\textsuperscript{a}, *John C.K. Cheung\textsuperscript{b,c}

\textsuperscript{a}Department of Civil and Environmental Engineering, The University of Melbourne, Australia
\textsuperscript{b}Department of Mechanical Engineering, Monash University, Victoria 3800, Australia
\textsuperscript{c}MEL Consultants Pty Ltd, 17 Kingston Street, East Malvern, Victoria 3145, Australia

ABSTRACT: A new turning rapid diffuser has been developed in this study using CFD analyses. Typically the diffuser angle should be less than 7 degrees to avoid flow separation. This angle requirement cannot be satisfied when the diffuser length is restricted, such as in the design of a wind tunnel circuit, by its overall dimension. A parametric investigation using ANSYS CFX 10 has found that the diffuser angle can be much larger without creating flow separation when the flow is diffused in a circular turning path. With this turning rapid diffuser, the flow has been shown to attach to the diffuser’s sidewalls even when the diffuser angle is as large as 12 degrees. Flow separation still occurs from the inner surface in the turning direction; however further CFD analyses have indicated that this separation can be avoided by the addition of guide vanes along the turning flow path.

KEYWORDS: Wind Tunnel Turning Rapid Diffuser CFD.

1 INTRODUCTION

Computational Fluid Dynamics (CFD) techniques have been developed in the past few decades to predict the flow characteristics by using computer-based numerical solutions of the fundamental fluid dynamics equations. Despite recent advances in computing capabilities and significant research in turbulence modelling, most current CFD models are yet unable to predict with sufficient accuracy the flow separation phenomenon, which is often encountered in Wind Engineering studies. However, in relatively simple flows, Leschziner [1] has demonstrated that eddy-viscosity models, constructed to give the correct level of shear stress, are able to yield adequate solutions. In particular, the Shear Stress Transport (SST) turbulence model can perform fairly well, especially in two dimensional flow situations.

Many numerical investigations have previously been carried out using various computational codes and turbulence modeling methods. Haskew and Sharif [2] have computed the stream-wise and cross-stream velocity distributions in turning pipe flow, using a relatively extensive developed Incompressible Navier-Stokes flow solver in 3 Dimensions with UPwinding code (INS3D-UP). Abe and Ohyu [3] investigated flow fields around flanged diffusers, using the non-linear eddy-viscosity model with some advanced closure approximation and modifications for the length-scale equation. Gullman-Strand et al. [4] carried out a numerical study of separated flow in a plane asymmetric diffuser with the Explicit Algebraic Reynolds Stress Model (EARSM). In spite of the complexity and also being not readily available, most computational formulations still present difficulties in accurately modeling the production and diffusion terms in the transport equation for the dissipation rate of turbulence in flow separated region in particular. The more recent Shear Stress Transport (SST) model is optimized for adverse pressure gradient flows with separation and thus gives highly accurate separation prediction as compared to the other previous computational methods. This SST model is readily available and included in the ANSYS CFX 10.0 software package [5].

In the process of designing a boundary layer wind tunnel for the Institute of Building Science and Technology (IBST), in Vietnam, one of the main difficulties as often found in wind tunnel circuit design in the past is the limitation of available space for the largest possible working section with the least possible loss of fan power. Barrett [6] used a short multi-cell diffuser to overcome this difficulty to be able to
achieve transition from the 0.5m by 1m high speed section to the 2m by 1m duct, noting the complex and time-consuming task in the diffuser construction. The proposed IBST closed-loop vertical return wind tunnel has a similar requirement for a 1 to 4 expansion ratio from a 2m by 2m working section to the larger 8m by 2m wide working section. The use of a multi-cell diffuser with an effective cone angle of 23° still requires a diffuser length of the order of 15m. This rapid diffusing transition is preferably to be installed in the turning section, where flows are more attached to the outer surface and guided with turning vanes, in order to maximize the effective straight tunnel fetch length for boundary layer generation. To provide a time-efficient and cost-effective evaluation of the flow analysis for this turning rapid diffuser, computational simulation of various operating conditions using commercially available computer software package is considered to be the best option.

This paper describes the use of CFD analysis in the development of a wide angle diffuser within a short turning flow path for the preliminary circuit design of the IBST wind tunnel in Vietnam. The CFD results are being validated with physical flow model measurements and some experimental data are also presented.

2 COMPUTATIONAL METHOD

The computer software CFD package, ANSYS CFX10.0 system, was used in the present flow analysis. There are four modules in the system, as shown in Figure 1, initially modeling the geometry with mesh of elements.

![Figure 1. ANSYS CFX System.](image)

The accuracy of the CFD model varies with the mesh quality and the type of elements, which can be classified in Figure 2. Results from this study have shown that the mesh modeling with Hexahedral elements would give similar flow pattern as measured from the scaled model experiments, while using Tetrahedral elements the computational model fails to predict the separation phenomenon.

![Figure 2. Type of elements used in CFX.](image)

Several CFD models have been developed in this study, but this paper only presents three typical models for comparison: the rapid turning diffuser using tetrahedral and hexahedral elements and the straight diffuser with the same diffuse angle using hexahedral elements. The diffusers were modeled as three dimensional tunnels with boundary walls (zero velocity on the wall surface), one inlet and one outlet. The fluid was modeled as air at 25°C with velocity of 20m/s and turbulent intensity of 5% distributed uniformly at inlet. The condition at the outlet was set at zero static pressure in order to ensure that all of the air is going out of the domain. The turbulence was transported using Shear Stress Transport (SST) turbulence model. The CFD models were solved in Pentium 4 PC, with 3.0GHz microprocessor and 1.5 Gb RAM. The simulation type of the models used was Steady State with conversion criteria as maximum of 100 iterations or Reynolds stress model (RSM) reaches $10^{-5}$. The modeling configurations and parameters of the models are given in Table 1 as follows:
Table 1. CFD Modeling configurations and parameters of diffuser from 2m×2m inlet to 2m×8m outlet

<table>
<thead>
<tr>
<th>Model No.</th>
<th>Description</th>
<th>Type of element</th>
<th>Number of nodes and elements</th>
<th>Running time</th>
</tr>
</thead>
</table>
| 1         | Turning diffuser totalangle=25° | Tetrahedral & Wedges | # Nodes: 560243  
# Tetra. Elements: 1844748  
# Wedges: 574920 | 8 hour and 6 minutes |
| 2         | Turning diffuser totalangle=25° | Hexahedral       | # Nodes: 556830  
# Elements: 538896 | 3 hour and 28 minutes |
| 3         | Straight diffuser totalangle=25° | Hexahedral       | # Nodes: 270680  
# Elements: 260800 | 1 hour and 37 minutes |

3 COMPUTATIONAL RESULTS

The flow separation regions for each model are shown from the iso-surfaces of longitudinal velocity in Figure 3. For the turning diffuser modeling with tetrahedral elements (model 1), the flow does not separate on the side wall on the turning part. However, when modeling with hexahedral elements (model 2) the flow separates near the side wall on the turning part. For both models 1 and 2 the flow separates at the inner wall. For the straight diffuser (model 3), the flow separates at the side walls.

![Model 1: using tetrahedral elements](image1)
![Model 2: using hexahedral elements](image2)
![Model 3: using hexahedral elements](image3)

Figure 3. Flow separation regions in a rapid turning diffuser and a straight diffuser.

By using the CFX software package, velocity streamlines can also be generated to show flow separation regions for various configurations of diffusers, as given in Figure 4. No separation is seen for the 10° straight diffuser while significant separation can be observed for the 25° straight diffuser. Also, flow separates from the inner surface of the 25° turning rapid diffuser, but is shown to be fairly uniform when two guide vanes were used.

![10° straight diffuser](image4)
![25° straight diffuser](image5)
![25° turning diffuser](image6)
![25° turning diffuser with guided vanes](image7)

Figure 4. Velocity streamlines showing separation regions for various diffuser configurations

---

751
4 EXPERIMENTAL RESULTS

A 1:5 scale model of the turning rapid diffuser was built, as shown in Figure 5. The flow was generated by an industrial fan and straightened with honey comb screens. Velocity and turbulent intensity of the flow were measured by using a TFI Cobra probe. The flow was uniformly distributed at the inlet, but the velocity at the outlet was measured to be much lower at inner wall than at outer wall. The flow separation phenomenon was observed on the inner wall. Further velocity measurements are being made on the turning rapid diffuser with guided vanes as suggested from the CFD analysis.

1:5 scale model of the turning rapid diffuser

Inlet velocity profile

Outlet velocity profile

Figure 5. Velocity streamlines showing separation regions for various diffuser configurations

5 CONCLUSIONS

CFD flow analysis by using ANSYS CFX software package with Shear Stress Transport (SST) turbulence model enables a relatively simple evaluation for flow through various configurations of rapid diffuser. Using hexahedral elements gives prediction of flow similar to the experimental measurement while the CFD model using tetrahedral elements fails to predict flow separation. A turning rapid diffuser with internal guided vanes is shown to provide uniform flow for a wind tunnel circuit with limited length.

6 ACKNOWLEDGEMENTS

This work is supported by the Ministry of Construction, Vietnam and all model measurements were carried out by the Institute of Building Science and Technology (IBST), Hanoi.

7 REFERENCES