Numerical study of effects of centre body on performance of a fan-discharge diffuser

Haikun Shi, Zhao Feng Tian, Peter Lanspeary, Richard Kelso

School of Mechanical Engineering, The University of Adelaide, South Australia 5005, Australia

Email Address: zhao.tian@adelaide.edu.au

ABSTRACT

This paper reports computational fluid dynamics (CFD) studies which test the effect of different centre bodies on the performance of Adelaide wind tunnel diffusers. The CFD method is validated by comparing simulations of annular diffusers with design charts from the literature. The static pressure coefficients from the CFD simulation and from design charts are different by not more than 7%. Two centre body designs were analysed to determine which performed better. The concentric centre body was designed to follow the offset (angle between centre body centre line and horizontal) of the diffuser, whereas the non-concentric centre body was axisymmetric and aligned with the axis of the upstream duct. The CFD results demonstrate the centreline orientation of centre body has a significant effect on diffuser performance for Adelaide wind tunnel. The concentric centre body with centreline is 3.5° from the horizontal gives a higher static pressure recovery, lower loss of total pressure drop coefficient and a more uniform outlet flow speed compared to those from diffuser with non-concentric centre body with horizontal centre line. The CFD results show that the length of the centre body also has significant effects on the diffuser performance. The diffuser with the shortest concentric centre body (1400 mm long) gives the highest static pressure recovery coefficient, the lowest loss of total pressure and the most uniform outlet flow speed, while the longest concentric centre body (2200 mm) gives worse diffuser performance compared to 1400mm centre body and 1800mm centre body. Furthermore, the diffuser without centre body gives the worst diffuser performance among all diffusers with centre body.

NOMENCLATURE

A outlet area of CFD model
C_{pr} static pressure recovery coefficient
h_2 annulus width
L diffuser length
Lc centre body length
L_d length of parallel duct downstream of diffuser
L_u length of parallel duct upstream of diffuser
K_t total pressure drop coefficient
R duct radius
P_1 average pressure of diffuser inlet
P_2 average pressure of diffuser outlet
P_{t1} total pressure of inlet of diffuser
P_{t2} total pressure of outlet of diffuser
ρ density
v flow velocity
$\bar{v}$ average flow velocity
$\emptyset$ angle between duct wall and duct axis
Δv no uniformity of flow speed at outlet of CFD model

INTRODUCTION

Wind tunnels have been used as a tool for investigation of fluid flows and product design in a wide range of engineering applications that can be found in mechanical engineering, aerospace engineering, sports engineering and civil engineering. With the rapid increase of computer power, computational fluid dynamics (CFD) has been used as a tool to investigate and improve the performance of wind tunnels. Moonen et al. (2006) developed a CFD model of a close-circuit wind tunnel. They found that CFD can generally reproduce the wind tunnel measurements of mean velocities with an error of 10% or less. Mahalakshmi et al. (2007) studied diffusers with three flow inlet conditions: a no wake flow, a shallow wake flow produced by a streamlined centre body, and a deep wake flow produced by a bluff centre body. They found the diffusers with a centre body have higher static pressure recovery coefficients than the diffusers without a centre body. Vinayak et al. (2011) numerically simulated the flow through honeycomb-screen combinations in a subsonic wind tunnel to determine the most suitable sizes of honeycombs and screens for the wind tunnel. Honeycombs and screens are used to reduce turbulence in the wind tunnel. The CFD results agree closely with experimental data and theoretical results.

The Adelaide wind tunnel (Figure 1) is the second largest wind tunnel in Australia, located at the Thebarton Campus of the University of Adelaide. In this wind tunnel, there is a diffuser on the discharge side of each of the six fans, as shown in Figure 1. There is currently no centre body in the current diffusers and therefore the purpose of this project is to report the effect of different centre bodies on the performance of the wind tunnel using CFD. The static pressure recovery coefficient, the energy loss coefficient and the velocity nonuniformity are used to assess the performance of a diffuser. According to Houghton et al. (2012), the static pressure recovery coefficient, C_{pr}, and the total pressure drop coefficient, K_t, are calculated as below:

$$C_{pr} = \frac{P_2 - P_1}{\frac{1}{2}\rho\bar{v}^2} \quad (1)$$

$$K_t = \frac{P_{t1} - P_{t2}}{\frac{1}{2}\rho\bar{v}^2} \quad (2)$$

where P_2 is the area averaged static pressure at the outlet of diffuser, for example plane 2 in Figure 2. P_1 is the area averaged static pressure at the inlet of diffuser, for example plane 1 in Figure 2. P_{t1} and P_{t2} are total pressure of inlet of diffuser and outlet of diffuser, respectively. $\bar{v}$ is the average velocity at a plane. The velocity nonuniformity Δv is the Root mean square (RMS) of the velocity difference between the node velocities and the average velocity at a plane.

In order to gain confidence in the CFD results, a validation case has been conducted in a benchmark case of circular annular diffusers in incompressible flow from ESDU 75026 (1975). ESDU 75026 is a design guide on the circular annular diffusers in incompressible flow published by Engineering Sciences Data Unit (ESDU) in the UK.

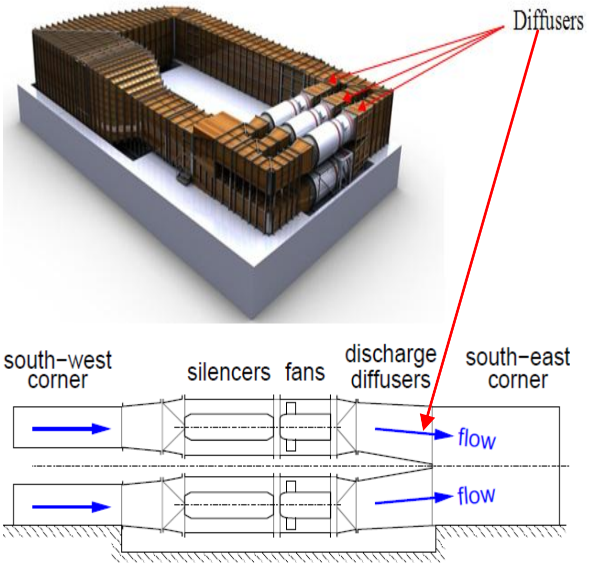


Figure 1: Adelaide wind tunnel and diffusers (Lanspeary & Kelso, 2012).

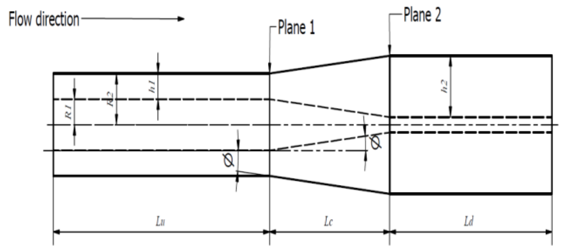


Figure 2: Geometry of the symmetrical circular annular diffuser model, $L_d/h_2 = 4$ (ESDU 75062 1975).

CFD MODEL DETAILS

In this study, the commercial CFD code, ANSYS/CFX 14.0, is used to simulate flows in the ESDU diffusers and the Adelaide wind tunnel diffusers. All geometries are generated by using ANSYS/Designmodeler 14.0 and mesh is generated using ANSYS/Meshing 14.0. The mesh sizes of all cases in the paper are given in Table 1 and Table 2.

For all wind tunnel diffuser models, the boundary conditions are the same. The inlet velocity is 30 m/s and the outlet relative static-pressure is 0 Pa. Shear stress transport (SST) model is used to model the turbulence. The convergence criteria for the air phase properties were set to 10^{-5} of the RMS.

In order to optimize the shape and dimension of the centre body for Adelaide wind tunnel, two parameters of the centre body were investigated in the preliminary analysis. The first one is the angle between the centre line of the centre body and the centre line of the diffuser.

The concentric centre body follows the offset of the inlet and the outlet. The centre body that does not follow the offset is called the non-concentric centre body. Figure 3 shows the concentric (Figure 3a) and non-concentric centre body (Figure 3b) models were created for analysis. In Figure 3a, the angle between the centre body centreline of the Adelaide wind tunnel Model 1 (concentric centre body) and the horizontal is 3.5° , which ensure the centre body follow the offset of inlet and outlet of the wind tunnel diffuser. In Figure 3b, the centre body centreline of the Adelaide wind tunnel Model 2 (non-concentric centre body) is parallel to the horizontal. The two centre bodies have the same centre body length (horizontal distance from the tip of the centre body to the inlet).

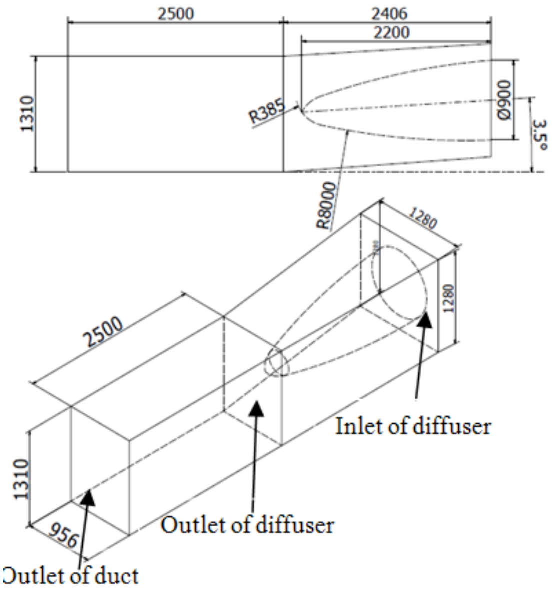
The second parameter is the length of the centre body. Another two concentric models (Adelaide wind tunnel Model 3 and Adelaide wind tunnel Model 4) with the same shape but different centre body length were simulated. Moreover, the wind tunnel diffuser model without a centre body (Adelaide wind tunnel Model 5) was also simulated so that the diffuser performance can be compared with those from diffusers with centre bodies. The parameters of different cases are listed in Table 3 and the configurations of Adelaide wind tunnel case 3, 4, and 5 are given in Figure 4.

Validation model number	Mesh nodes	L/h_1 ratio	Ø
1	466395	$L/h_1=5$	5°
2	474812	$L/h_1=8$	5°
3	456028	$L/h_1=10$	5°

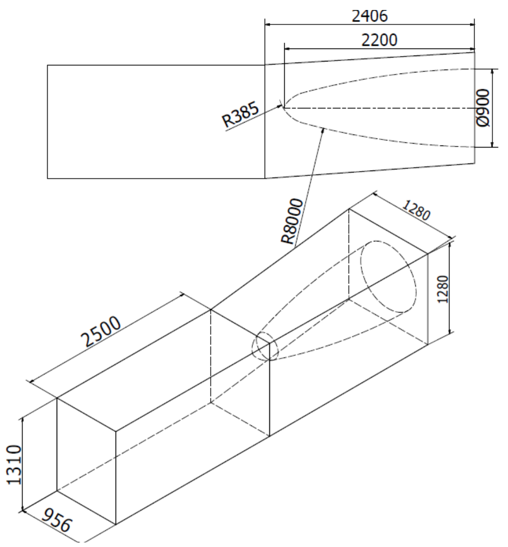
Table 1: Mesh nodes and parameters of the validation cases

Adelaide wind tunnel model number	Mesh nodes	Offset angle	Centre body length(mm)
1	468148	3.5°	2200
2	470881	0°	2200
3	465189	3.5°	1800
4	465960	3.5°	1400
5	435889	N/A	0

Table 2: Mesh nodes and parameters of the Adelaide wind tunnel cases

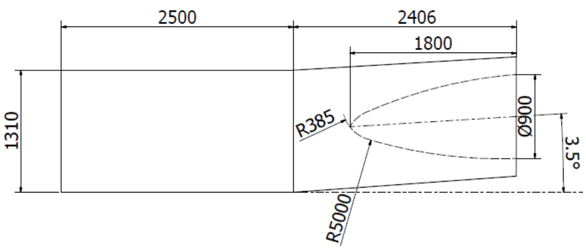


(a) Model 1-2200mm concentric centre body

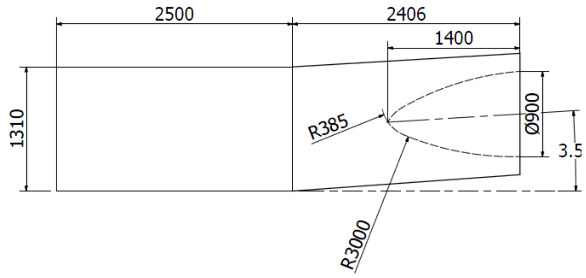


(b) Model 2-2200mm non-concentric centre body

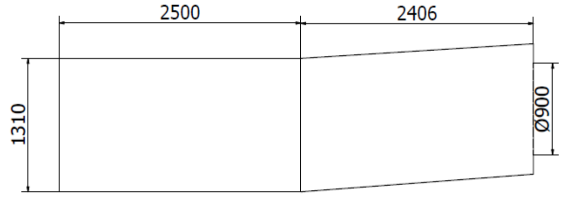
Figure 3: Model 1 and model 2 of Adelaide wind tunnel (unit mm).



(a) Model 3-1800mm concentric centre body

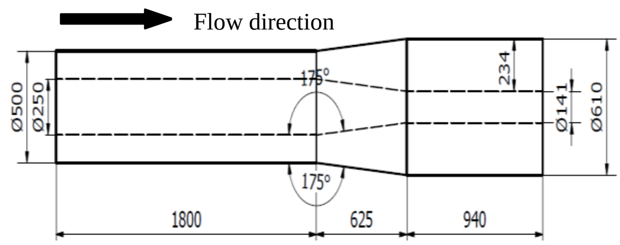


(b) Model 4-1400mm concentric centre body

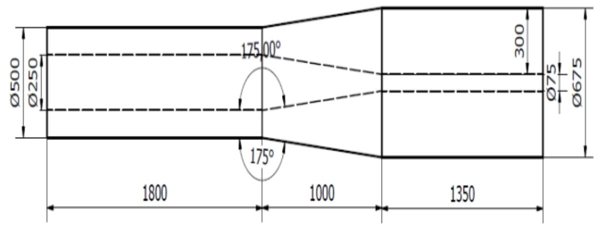


(c) Model 5-no centre body

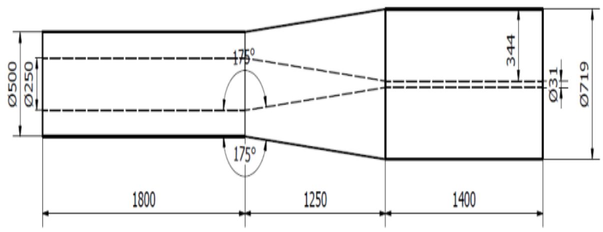
Figure 4: Model 3, 4 and 5 of Adelaide wind tunnel diffuser (unit mm).



Validation case 1



Validation case 2



Validation case 3

Figure 5: Geometry and dimensions of three validation models.

VALIDATION CASES

In order to estimate the accuracy of the CFD results, flows in three axi-symmetric circular annular diffusers have been simulated and static pressure recovery coefficients are compared with estimates from ESDU 75026. The geometry and dimensions of these three models are given in Figure 5. In the models, the lengths of the diffuser are changed while the cone angles remain constant. As a result, duct diameters at the diffuser outlets are also changed.

Validation case	C_{pr} (CFD)	C_{pr} (ESDU)	Difference (%)
1	0.57	0.61	6.7
2	0.66	0.67	1.5
3	0.76	0.72	5.6

Table 3: Comparison of predicted static pressure recovery coefficient (C_{pr}) with e of ESDU 75026.

Table 3 compares the predicted pressure recovery coefficients of three cases against the data obtained from ESDU 75026. It can be seen that the static pressure recovery coefficients from CFD simulation agree well with the ESDU 75026 estimate results. In all cases, the difference between the CFD results and ESDU 75026 is less than 7%, which indicates the SST model is appropriate for the flows in the diffusers with centre body.

RESULTS AND DISCUSSIONS

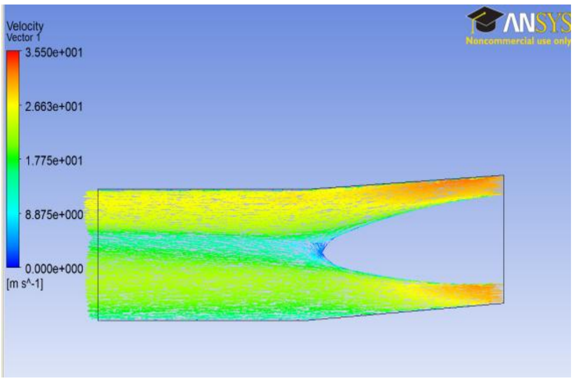
Table 4 shows static pressure recovery coefficients, total pressure drop coefficients and nonuniformity of flow speed of exit plane of duct downstream of Adelaide wind tunnel Model 1 and Model 2. The pressure difference used in the calculation refers to the pressure difference between the inlet of the diffuser and the outlet of the diffuser which are shown in Figure 3. The model with the concentric centre body has higher static pressure, lower total pressure drop compared to results from model 2 with the non-concentric centre body. Furthermore, the outlet flow of the model with the concentric centre body is more uniform than that of the model with the non-concentric centre body. Hence, the model with the concentric centre body has better performance. Figure 6 (b) shows the poorer performance of the non-concentric is due to a region of high velocity between the top of the centre body and roof of the diffuser. Whether the performance of the diffuser would be improved by further adjustment of the angle between the centre body centre line and the horizontal should be investigated in the future.

Model number	C_{pr}	K_t	Δv
Adelaide wind tunnel Model 1 (concentric)	0.258	0.072	3.2534
Adelaide wind tunnel Model 2 (non-concentric)	0.225	0.097	4.2888

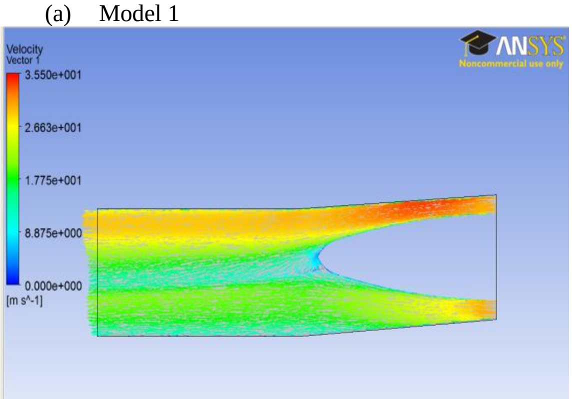
Table 4: Static pressure recovery coefficients (C_{pr}) and loss coefficients (K_t) and flow speed no uniformity (Δv) for model 1 and model 2.

The current results demonstrate that the concentric centre body has better performance than the performance of diffuser with no-concentric centre body. As a consequence, the concentric model is used for the centre-body length investigation. Another three models are generated to investigate the effects of centre body length on the diffuser performance. Details of the models can be found in Figure 4 and Table 3.

Table 5 shows the static pressure recovery coefficients, total pressure drop coefficients and nonuniformity of the flow speed at exit plane of the duct downstream of models 3, 4 and 5. Figure 7 shows velocity vector graphs of the centre plane of these models. From Table 5, the shortest centre body (Model 4) has the highest static pressure recovery coefficient and the lowest total pressure drop coefficient. It should be noticed that the shortest centre body model also provides the most uniform outlet flow velocity; however, the difference between this and the results of Model 3 and 4 are quite small. The two shorter centre body Models 3 and 4 give very similar and superior diffuser performance. Also, the model without any centre body has the highest total pressure drop and the lowest static pressure recovery coefficient among all the models and the worst outlet velocity uniformity. This occurs because the absence of centre body leading to a large recirculation zone which is shown in the Figure 7(C). However, the outlet flow velocity uniformity of Model 2 is worse than that of Model 5 even if the non-concentric gives higher static pressure recovery coefficient and lower pressure drop.



(a) Model 1

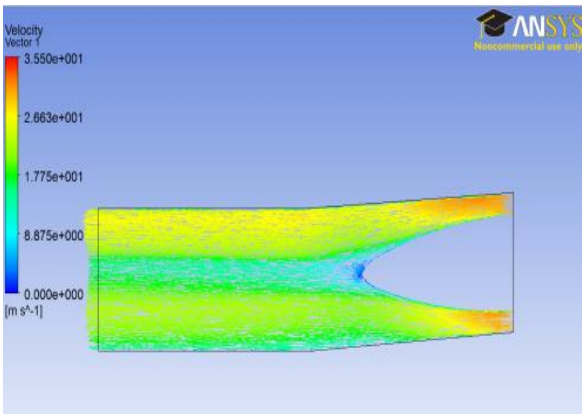


(b) Model 2

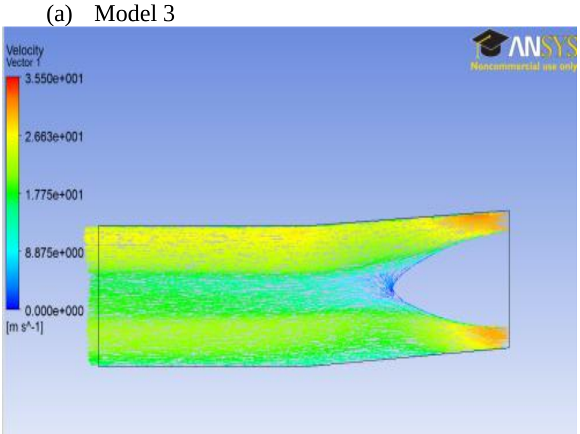
Figure 6: velocity vector graphs of centre plan of diffuser model with concentric and non-concentric centrebody respectively.

Model	C_{pr}	K_t	Δv
3	0.272	0.063	3.0581
4	0.277	0.058	2.9176
5	0.149	0.159	3.7652

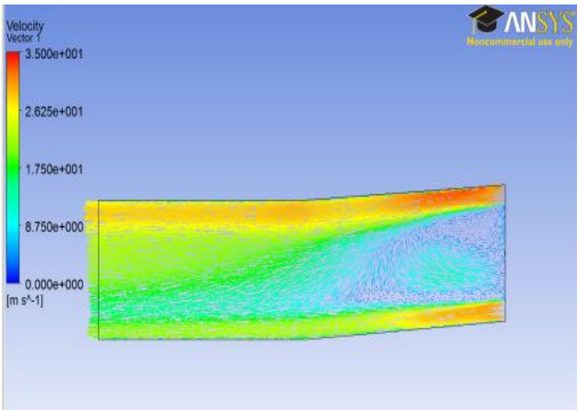
Table 4: static pressure recovery coefficient, total pressure drop coefficient and RMS of model 1, 3, 4



(a) Model 3



(b) Model 4



(c) Model 5

Figure 7: velocity vector graphs of centre plan of Adelaide wind tunnel Model 3, 4 and 5.

CONCLSION

The aim of work presented in this paper is to improve Adelaide wind tunnel diffuser performance by optimizing the shape and size of centre body. As a consequence, 5 wind tunnel diffuser models were created. The comparison between models 1 and 2 indicates the model with the concentric centre body has better performance than the model with the non-concentric centre body. Future work will include refinements to the centre body angle in order to achieve further improvements. The comparison between models 1, 3, 4 and 5 demonstrates that the centre body size also has a significant influence on the diffuser performance. The results demonstrate that the centre body length, two models with shorter concentric centre bodies with lengths of 1400mm and 1800mm respectively, provide similar and better diffuser performance compared to others.

REFERENCES

ENGINEERING SCIENCES DATA UNIT, (1975), *Performance of circular annular diffusers in incompressible flow; Item No.75026*. Obtained from Engineering Sciences Date Unit, 251-259 Reent St. LONDON W1R7AD.

HOUGHTON E., CARPENTER P., COLLICOTT S. and VALENTINE D., (2012), “Aerodynamics for Engineering Students”, *Elsevier*, UK.

LANSPEARY P. and KELSO R., (2012), “On the design of a recently constructed 285 m^3/s wind tunnel at the University of Adelaide”, *18th Australasian Fluid Mechanics Conference Launceston*, Australia.

MAHALAKSHMI N.V., KRITTHIGA G., SANDHYA S., VIKRAMAN J. and Ganesan, V., (2007), “Experimental investigations of flow through concial diffusers with and without wake type velocity distortions at inlet”, *Experimental Thermal and Fluid Science*, No.32, pp. 133-157.

MOOEN P., BLOCKEN B., ROELS S. and CARMELIET, J., (2006), “Numerical modeling of the flow in a closed-circuit low-speed wind tunnel”, *Journal of Wind Engineering and Industrial Aerodynamics*, No. 94, pp. 699-723.

VINAYAK K., NIANJAN S. and SANDIP D.C., (2011), “Simulation of honeycomb-screen combinations for turbulence management in a subsonic wind tunnel”, *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 99, pp. 37-45.