15th Australasian Wind Engineering Society (AWES) Workshop
Sydney, Australia
23-24 February, 2012

CFD Simulations of the New University of Sydney Boundary Layer Closed Circuit Wind Tunnel

A. Bertholds[1], S. Cochard[2] and D. F. Fletcher[2]

[1]Faculty of Science and Technology
Uppsala Univeristy, Uppsala, Sweden

[2]Faculty of Engineering & IT
The University of Sydney, Sydney, NSW 2006, Australia

Abstract

Computational Fluid Dynamics (CFD) simulations were performed to evaluated the concept design of the new University of Sydney closed circuit boundary layer wind tunnel. Even if each section of the tunnel could be optimised individually, the objective of the project was to simulate the complete wind tunnel and optimise the overall design. The goal was to design a wind tunnel that gives a uniform flow in the test section while minimising the pressure loss over the whole tunnel for a given flow velocity. The simulations show that a nearly uniform flow can be obtained by using several flow-improving components such as turning vanes, splitters, screens, a honeycomb and a settling chamber.

Introduction

In 2012, the School of Civil Engineering at the University of Sydney will build a new closed circuit boundary layer wind tunnel. The wind tunnel should have an optimal design within the constraints imposed by the project's budget and the space where it will be located. The brief for the new tunnel was to have a smooth and uniform flow in the test section with a flow velocity of 30 m/s, while minimising the pressure loss The objective of this project was to create a computational fluid dynamics (CFD) model of the tunnel, that could be used to optimise the flow and to evaluate the concept design of the tunnel. The model was constructed so that it could be divided into several sections. This would make it possible to modify just one or a few sections at a time, and then reassemble all the sections to see the influence of the modifications on the flow in the full tunnel.

The concept design simulated was mainly based on the Sydney University - Wind Tunnel Performance Brief by Flay (2011) and suggestions given in Mehta and Bradshaw (1979) and Barlow et al (1999).

To obtain a uniform flow in a wind tunnel, several flow improving components are needed, such as vanes, a settling chamber, honeycombs and screens.

Turning vanes are used in the corners of the tunnel to prevent flow separation by forcing the flow into a desired path through the corner (Barlow et al, 1999). A turning vane is commonly designed as a quarter circle with an extension of the trailing edge, which forms a 0^o angle to the surrounding walls, and with a 4^o angle of attack to the incoming flow (Mehta and Bradshaw, 1979; Barlow et al, 1999). The ratio of the gap between the vanes to their chords should not exceed 0.25.

A settling chamber is a section of the tunnel where the duct area is locally increased. The flow decelerates due to this expansion, which makes the settling chamber a good location for a heat exchanger to remove the heat produced by the fan while minimising the pressure losses. The flow accelerates when it leaves the settling chamber through a contraction, which reduces the level of turbulence. The flow leaving the settling chamber is typically aligned by using a honeycomb.

The sudden expansion the flow experiences when it enters the settling chamber causes unwanted flow separation. Barlow et al (1999) suggest the use of screens with a loss coefficient between 0.5 and 0.8, to reduce the flow separation. A screen evens out differences in the flow velocity by imposing a pressure drop proportional to the square of the flow velocity and therefore prevents flow separation.

The CFD model of the tunnel was used to investigate how these components should be used to obtain the smooth and uniform flow in the test section.

Method

The CFD model of the tunnel is shown in Figure 1, in which the air flows clockwise. In the CFD model, the flow starts from the inlet before entering a circular-to-rectangular expansion followed by a long, slowly diverging duct and the first of two corners. In the corners, the turning vanes, designed after the suggestions given in Mehta and Bradshaw (1979) and Barlow et al (1999), force the flow through the corner. The radius of the turning vanes was 40 cm, the same as for the corner's fillet, and the trailing edge was 15 cm long. The gap to chord ratio was approximately 0.25.

After the two first corners, the flow enters the settling chamber through a divergent section. In the tunnel a heat exchanger will be situated in the settling chamber but it was omitted at this stage. Apart from using screens to prevent flow separation in the divergent section, as suggested by Mehta and Bradshaw (1979) and Barlow et al (1999), splitters forming a tubular square pattern were inserted in the divergent section. These splitters spread the air flow evenly into the settling chamber and prevent large recirculation. The splitterss are necessary due to the relatively short length of the divergent section.

The flow leaves the settling chamber through the convergent section before entering the long straight duct in which the test section with the turn-table is located. After the test section, the flow goes through the last two corners, also equipped with turning vanes, and through a converging rectangular-to-circular duct before it leaves the computational domain through the outlet. The tunnel width reduction after the second last corner is constrained by the space available.

The screens and the honeycomb in the tunnel were modelled as porous materials in CFX. The real screens will be a couple of millimetres thick, but in order to get an accurate simulation they were modelled with a thickness of 20 cm so that sufficient many mesh elements could fit in them.

The fan was modelled as two separate parts, the inlet and the outlet, which made the flow leaving the fan independent of the flow entering it. This configuration maintained a flow speed of 30 m/s in the test section when different tunnel designs were

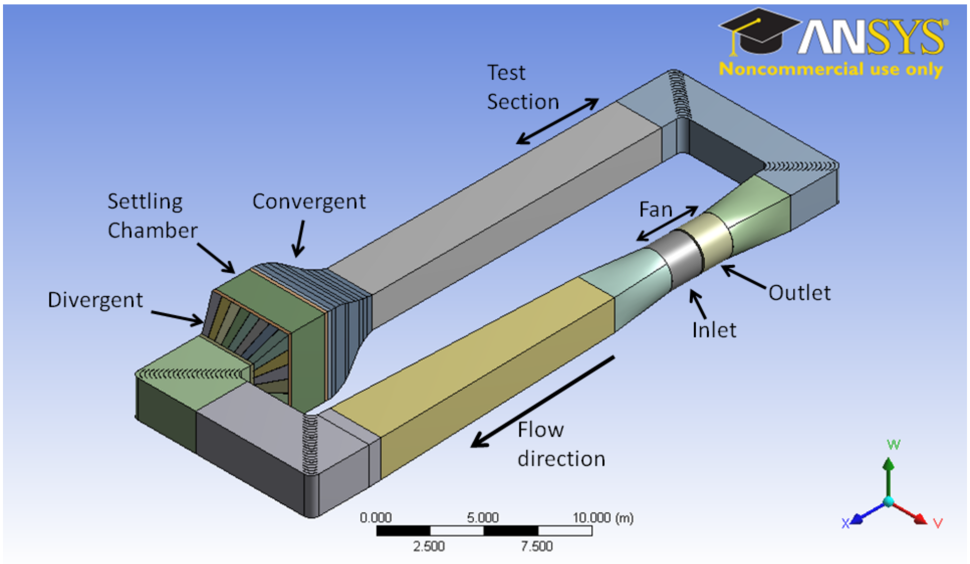

Figure 1: The model of the wind tunnel

tested, for example when the number of screens were changed and gave an overall pressure loss. A swirl component of velocity and high turbulence level was imposed at the inlet to simulate the flow out of the fan.

The CFD simulations were made with ANSYS-CFX 13, which uses a finite volume method to solve the mass, momentum and energy equations ANSYS (2010). The velocity calculations are made with a second order bounded differencing scheme while a first order upwind scheme is implemented for the convective terms in the turbulence equations. For all the diffusive terms, a second order scheme is used.

The Shear Stress Transport (SST) model was used to model the turbulence. By combining the $k-\varepsilon$ and the $k-\omega$ models with a blending function, Menter (1994) proposed the SST model in 1994 that gives more accurate flow separation predictions. The $k-\varepsilon$ model is used in the free shear region, while the $k-\omega$ model is used in the boundary layer near the walls.

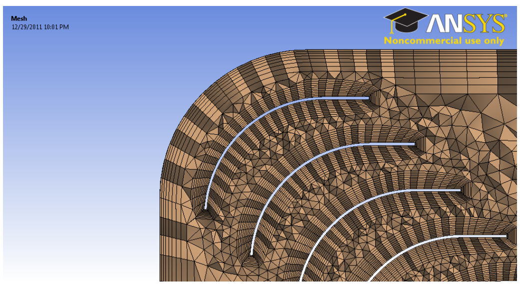

Figure 2: A close up of the mesh around the corner turning vanes and at the corner's wall. The flow enters from the bottom of the figure.

The computational mesh consisted of both tetrahedral and hexahedral elements, summing up to 15.9 million elements. The hexahedral elements were used in all sweepable parts (i.e, the divergent section, the settling chamber, the two long ducts of the tunnel and the fan). Tetrahedral elements were used in all other irregular parts (i.e. the corners, the convergent section, the circular-to-rectangular divergent section and rectangular-to-circular convergent section). A 15 cm thick inflation mesh, consisting of 10 layers, was used on the walls of the tunnel. The turning vanes in the corners and the splitters in the divergent section also had an inflation layer, it consisted of 10 layers with a first layer height of 0.5 cm. Figure **??** shows the inflation around the turning vanes in one of the corners. The coarsest element size allowed was 23 cm. In the settling chamber and in the convergent section a body-sizing of 15 cm was used to give a better resolution of the recirculation zones in the settling chamber. Since the flow changes rapidly around the corner turning vanes, they were given a face-sizing of 2.5 cm.

Results

Many different configurations were tested to obtain an optimal flow. The number of screens and their location was varied, as were their loss coefficients and porosity settings. Simulations with the splitters in the divergent section replaced by several screens were also made. However, this gave a significant decrease in flow quality. The corner turning vanes gave a significant improvement of the flow. Not only did they prevent distortion of the flow in the corners and reduced the pressure loss in the tunnel, but they also reduced most of the swirl introduced by the fan.

The configuration giving the best flow had three screens and one honeycomb, together with the use of splitters in the divergent section. The two first screens were located directly before and after the divergent section. The third screen was located at the end of the settling chamber, adjacent to the honeycomb. All screens had a loss coefficient of 0.5 and a volume porosity of 50%. The honeycomb, located just before the convergent section, also had a loss coefficient of 0.5 and a volume porosity of

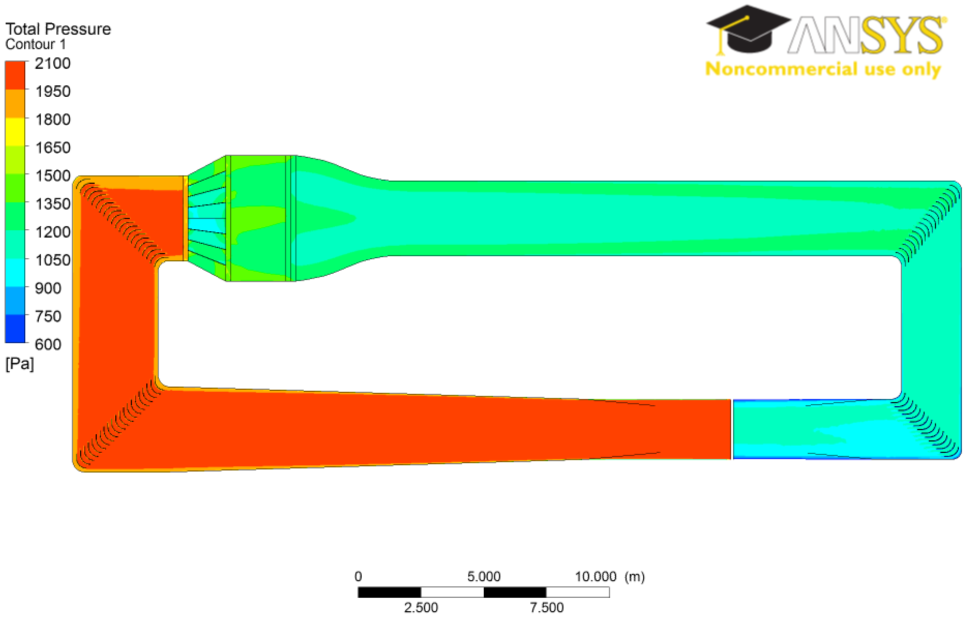

Figure 3: Contour plot of the total pressure

75%, meaning that air could flow through 75% of the honeycomb's volume. A flow speed of 30 m/s in the test section was obtained when the flow speed out of the fan was 40 m/s. The loss in total pressure was 1.3 kPa. A contour plot off the loss in total pressure is shown in Figure 3. The biggest pressure loss is found, as expected, over the first screen where the velocity is maximum (see Figure 3).

Figure 4 shows the streamlines in the whole tunnel while Figure 5 shows the streamlines in the settling chamber. There is some recirculation present in the settling chamber, indicated by the low-speed streamlines.

Figure 6, show the tangential component of the velocity in the test section. The maximum cross-wise velocity in test-section is 0.5 m/s. Even if the cross-wise velocity is relatively small compared with the 30 m/s of the flow, it needs to be reduced. In an ideal case all the components of the velocity, except the stream-wise one, should be equal to zero. It is interesting to note that the asymmetry of the flow in the test section is induced by the downstream corner and not by the upstream convergent section or duct. This emphasises the necessity to simulate the complete tunnel.

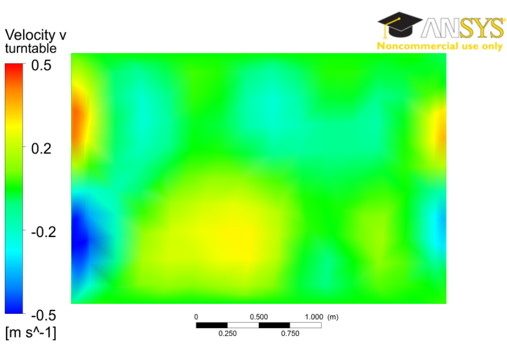

Figure 6: Contour plot of the tangential component of the velocity at the turn-table.

Discussion

The results show that the turning vanes in the corners and the splitters in the divergent section are essential in obtaining a uniform flow in the test section and to keep the pressure loss low. Using screens instead of splitters in the divergent section increased the pressure loss significantly without giving an improved flow. However, the flow quality was not good enough if only the splitters were used so a combination of screens and splitters is necessary.

The flow quality over the turn-table could be optimised by increasing the size and length of the settling chamber. Unfortunately, due to space restriction, these two solutions are not feasible. A better arrangement of the splitters could improve the flow in the settling chamber and guarantee a more uniform flow in the settling chamber and therefore in the convergent section and, as a result, in the test section.

When optimising each part separately, it was noted that the flow in the test section was more homogeneous and symmetric when the last two corners leading to the fan were removed, i.e. when the tunnel outlet was directly behind the test section. This indicates that the flow at the test section is affected by the downstream corners. A possible solution to reduce its influence on the test section, is to put a screen before the downstream corner. It is not possible to move the corners further downstream due to the limited construction space.

The recirculation in the settling chamber causes the flow to be unsteady and hence there were no steady-state numerical solution to this problem. The numerical instability was also indicated by the slightly oscillating root mean square residuals. However, the root mean square residuals went below 10^{-5} and the velocity at the test section was nearly constant. Therefore, the results still give a good indication of the flow behaviour in the tunnel, even though the location and the nature of the recirculation in the settling chamber will be different from what can be seen in Figures 4 and 5.

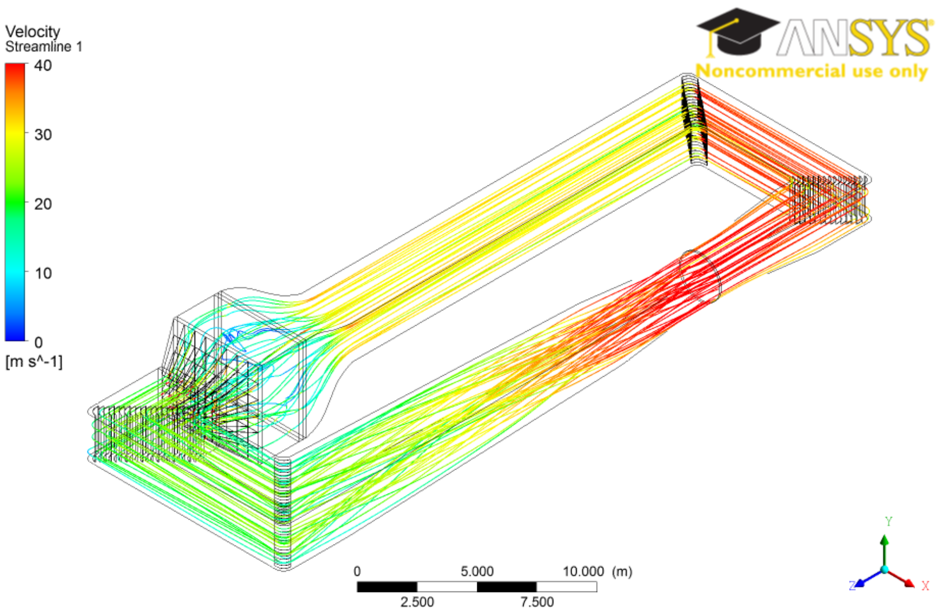

Figure 4: Velocity streamlines in the tunnel

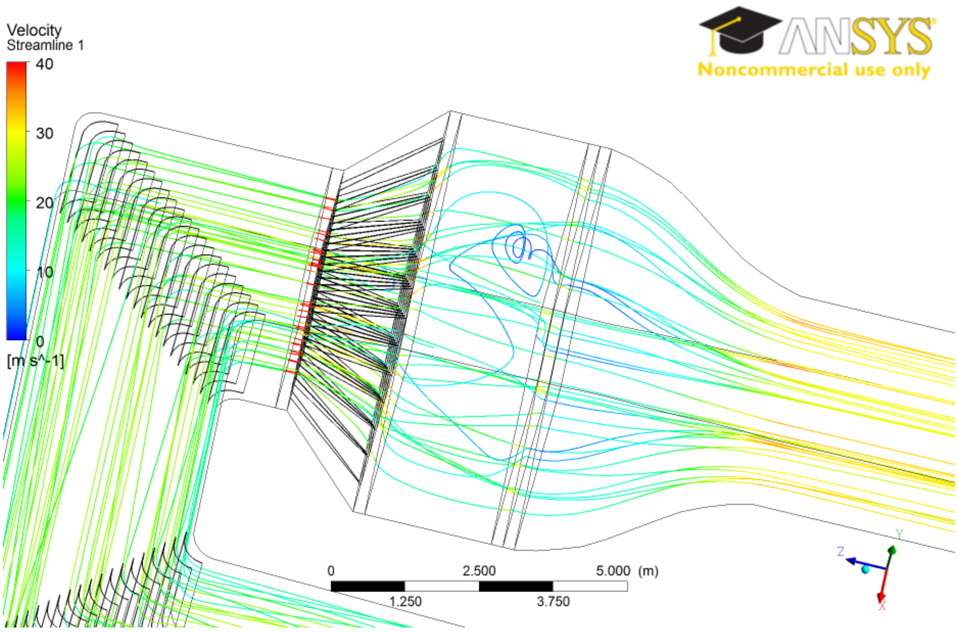

Figure 5: A close up of the velocity streamlines in the settling chamber. Note the small recirculations.

Conclusions

The CFD model made it possible to optimise each section of the tunnel individually and to see the effect of the optimisation on the full tunnel. A nearly uniform flow with a speed of 30 m/s was obtained in the test section. A fan that can produce a flow speed of 40 m/s and overcome a pressure loss of approximately 1.3 kPa is required. The simulations confirmed that the concept design is good even though further detailed design work needs to be performed.

*

References

ANSYS (2010) CFX users manual. URL www.ansys.com

Barlow J, Rae W, Pope A (1999) Low-Speed Wind Tunnel Testing. John Wiley and Sons Inc.

Flay R (2011) The sydney university - wind tunnel performance brief. Tech. rep., The University of Auckland

Mehta RD, Bradshaw P (1979) Technical notes - design rules for small low speed wind tunnels. The Aeronautical Journal of the Royal Aeronautical Society 73:443–449

Menter F (1994) Two-equation eddy-viscosity turbulence models for engineering applications. AIAA 32(8):1598–1605