A Computational Fluid Dynamic Study for Temperature Uniformity at a Wind Tunnel Outlet with Effects of Heat Transfer along Tunnel Walls

Li Song\textsuperscript{1,2,a}, Chen XiaoBing\textsuperscript{3}, Huang ZhiLong\textsuperscript{1,2} and Liao DaXiong\textsuperscript{1}

\textsuperscript{1} State Key Laboratory of Aerodynamics, China Aerodynamics Research and Development Center, Mianyang Sichuan 621000, P R China
\textsuperscript{2} Facility Design and Instrumentation Institute, China Aerodynamics Research and Development Center, Mianyang Sichuan 621000, P R China
\textsuperscript{3} Advanced Analysis Group, Advisian Singapore (WorleyParsons Group), #13-03 TripleOne Somerset, 111 Somerset Road, Singapore, 238164

\textsuperscript{a} Corresponding author: lisonic@foxmail.com

Abstract. A Computational Fluid Dynamics (CFD) Modelling was carried out to investigate temperature uniformity at a wind tunnel outlet region to benefit its design and construction. Low temperate (110K) nitrogen entered the wind tunnel at inlet, passed through two turning area with 18 turning vanes and went into expansion session, stable session and congestion session, finally towards outlet discharge nozzle session. The model considered static nitrogen isolation layer in physical model and three more isolation layer outside the tunnel as well as possible heat transfer from ambient environment. Two case scenario for supersonic and subsonic at outlet (Mach Number 0.9 and 1.3) were conducted with a uniform temperature imposed at inlet, as well a 3rd case scenario (Mach Number 0.9) with a non-uniform temperature distribution at inlet. Existence of porous zone (to model honeycomb session) and porous interfaces (to model damping interfaces) at the stable session were also included. Contours of temperature and velocity on representative cross sections were presented, and effects of Mach number, total pressure and initial temperature distribution at inlet on temperature uniformity at outlet were discussed. Based on the numerical results, improvement of tunnel geometry for better flow uniformity at outlet regions were also suggested.

1. Introduction
Wind tunnel has been a well-established discipline and it is a mature tool used in aerodynamic research [1-2]. Large-scale cryogenic wind tunnels, such as ETW and NTF, are the best ground testing facilities to obtain aircraft flow characteristics in the real flight conditions [3]. Design and testing of such wind tunnels become an important task before construction of these tunnels, in which CFD plays an increasingly important role [4-6]. With the development of modern technology, it is possible to do full-scale CFD modelling with high performance computers, which will reduce demand and cost for wind tunnel physical testing [7-9].

To assess temperature uniformity at a wind tunnel outlet region and finally to benefit its design and construction, a Computational Fluid Dynamics Modelling was carried out to investigate temperature uniformity at a wind tunnel outlet region. The target was to find effects of incoming flow at inlet (low and high Mach number, total pressure, and uniform or non-uniform temperature distributions) as well
as four layer isolation materials outside the tunnel wall. Possible heat transfer from ambient environment was also considered.

CFD simulation results were presented with contours of temperature and velocity on representative cross sections. Based on the numerical results, improvement of tunnel geometry for better flow uniformity at outlet regions were also suggested.

The High-performance computing for CFD model construction, Meshing, set-up and running was finished with Ansys WorkBench.

2. Model and Boundary Condition

3D geometry was imported into Ansys SpaceClaim and simplified into a half due to its symmetry property at the center interface (Figure 1). The structure was defeatured or re-organized to fit requirements for meshing. There was slight geometry difference for subsonic (Ma=0.9) and supersonic (Ma=1.3) at the discharge nozzle outlet region: one was flat extended (Ma=0.9), and another one was enlarged towards exit (Ma=1.3).

![Figure 1. Simplified model for tunnel fluid domain and outer isolation material domain.](image)

Another three layer of isolation materials was imposed as shell boundary with their thickness values to consider heat transfer in radial and axial directions (Table 1).

| Material          | Location                | Conductivity(w/m/k) | Remark          |
|-------------------|-------------------------|---------------------|-----------------|
| Aluminium         | 1<sup>st</sup> layer touching with tunnel fluid | 120.0               | Shell boundary  |
| Static Nitrogen   | 2<sup>nd</sup> layer    | 0.0242              | Physical model  |
| Isolation material| 3<sup>rd</sup> layer    | 0.037               | Shell boundary  |
| Steel             | Last layer (4<sup>th</sup>) to environment | 16.0                | Shell boundary  |

2.1. Meshing

A mixed type of mesh was used for both fluid and solid regions, totally 21.5 million cells (Figure 2). The mesh was composed of hexahedral and tetrahedral cell types in different tunnel sessions, and 10-15 prismatic cell along wall boundary as well as turning vane walls at the two turning corners. A minimum three-layer cells were imposed for the solid domain to consider heat transfer with 2<sup>nd</sup> order accuracy.
Figure 2. Overall mesh illustration for both solid and fluid domain.

2.2. Boundary condition
An overview of boundary conditions was illustrated in Figure 3. For the three case scenario, inlet conditions for total temperature and pressure were adjusted to meet their requirements at outlet (Table 2). For case 1 and case 2, a uniform temperature field was imposed at inlet; for case 3, a non-uniform temperature field was imposed:

\[ T = 2.5 \times \sin(2\pi \times y) + 110 \]  \hspace{1cm} (1)

Figure 3. Set-up for boundary conditions.

To consider existence of honeycomb structure at the stable session, one porous media zone was imposed with a pressure coefficient set up for a pressure jump across it around 500 pascal. To consider effects of damping nets there, 7 layers of porous resistance interface were imposed as porous-jump interface condition.

Table 2. Three case outlet scenario.

|         | Total temperature (K) | Total pressure (Pa) | Mach number | Temperature at inlet |
|---------|-----------------------|--------------------|-------------|----------------------|
| Case 1  | 110                   | 115                | 1.3         | uniform              |
| Case 2  | 110                   | 450                | 0.9         | uniform              |
| Case 3  | 110                   | 450                | 0.9         | sinusoidal           |

Heat transfer from ambient environment through tunnel walls and isolation materials was considered with a heat transfer coefficient \( h \) :

\[ Q = h \times (T - T_{\text{ambient}}) \]  \hspace{1cm} (2)
where the environmental temperature $T_{\text{ambient}} = 25 \, ^\circ\text{C}$.

An ideal-gas law physical model was applied for the fluid inside and a k-epsilon turbulent fluid model with standard wall function treatment was implemented. The numerical model was set-up and the simulations were running using Ansys Fluent 18.2. Due to the model size, parallel simulations over 8 cores were implemented and each case scenario took around 24-48 computer hours to get converged solutions.

3. Results and Discussion

Figure 4 to Figure 6 show velocity streamlines in the tunnel, velocity and temperature contours along the symmetry bottom surface for case 2 $\text{Ma} = 0.9$. Velocity and its related Mach number follow what expected: velocity develops and become uniform after passing by porous media zone and porous interfaces at the stable tunnel session. After entering the discharge area, due to decrease of tunnel region, the velocity increases and temperature decreases until outlet location (Figure 6).

Figure 4. Overall streamlines for case 2 ($\text{Ma} = 0.9$).

Figure 5. Overall velocity contour for case 2 ($\text{Ma} = 0.9$).

Figure 6. Overall temperature contour for case 2 ($\text{Ma} = 0.9$).
It is discovered that due to the existence of 2nd turning corner, there are fluid separation and a big circulation in the downstream (Figure 5), before the porous honeycomb zone. It is also noted there is a slightly small circulation at the 1st turning corner (circled in red in Figure 5). These two circulation may have negative impact on the flow and temperature uniformity in the downstream.

Figure 7. Overall heat flux contour for case 2 (Ma = 0.9).

Figure 7 presents wall heat flux distribution for case 2. It is noted that normally heat flux with thicker isolation material layers close to the outlet region is smaller compared to those with thinner isolation material layers in the upper stream. A maximum heat flux of 28.92 w/m² is found, which is within the design criteria requirement.

Figure 8. Overall temperature contour for case 3 (non-uniform temperature at inlet, Ma = 0.9).

Figure 8 shows the temperature contour for Case 3 with non-uniform temperature imposed at inlet. It is noted that temperature variation exists along the flow downstream, and after passing the porous media (honeycomb) and porous-jump interface (damping interface), the temperature field becomes more uniform. This shows that existences of honeycomb region and porous interface are able to help to improve flow and temperature uniformity.

Figure 9 show temperature difference contour \((T-T_{avg})\) at outlet surface (80% of the outlet surface, without boundary layer) for the 3 cases. It is found that when Mach number is smaller (Ma=0.9, case 2), the temperature variation at outlet is smallest \((\delta t = 0.1168 \text{ K})\); when Mach number is larger (Ma=1.3, case 1), the temperature variation at outlet also increases \((\delta t = 0.5219 \text{ K})\). However, a non-uniform temperature at inlet (Ma=0.9, case 3) gives a highest temperature variation at outlet \((\delta t = 0.8766 \text{ K})\). This variation value exceeds the maximum allowable temperature non-uniformity of 0.5 K. This can be improved by changes of structure at the 1st and 2nd turning corner to have smaller circulations, or adjusting the locations of porous honeycomb and porous damping interfaces.
4 Conclusion

CFD Modelling analysis was carried out to investigate temperature uniformity at a wind tunnel outlet region to benefit its design and construction. Low temperate (110K) nitrogen entered the wind tunnel at inlet, passed through two turning area with 18 turning vanes and went into expansion session, stable session and congestion session, finally towards outlet discharge nozzle session. The model considered static nitrogen isolation layer in physical model and three more isolation layer outside the tunnel as well as possible heat transfer from ambient environment. Two case scenario for supersonic and subsonic at outlet (Mach Number 0.9 and 1.3) were conducted with a uniform temperature imposed at inlet, as well a 3rd case scenario (Mach Number 0.9) with a non-uniform temperature distribution at inlet. Existence of porous zone (to model honeycomb session) and porous interfaces (to model damping interfaces) at the stable session were also included in the model.

The results showed that two circulation zones were found at the 1st and 2nd turning corner, which may impact flow and temperature uniformity downstream. It was also proved that existences of porous honeycomb and porous damping interface were capable to improve temperature uniformity at the outlet regions. However, it was suggested that improvement of structure of turning vanes at the 1st turning corner, and 2nd turning corners, location changes of honeycomb and damping interfaces might be necessary to have smaller circulation zones so that the impact on downstream is minimum.

References
[1] Wu Fenglin, Wang Zhenyu. The principles of wind tunnel design[M]. Beijing: Beijing Publishing House, 1985.

[2] J.E. Cermak. Wind-tunnel development and trends in applications to civil engineering[J]. Journal of Wind Engineering and Industrial Aerodynamics, 2003, 91(3):355-370.

[3] Liao Daxiong, Huang Zhilong, Chen Zhenhua, Tang Gengsheng. Review on large-scale cryogenic wind tunnel and key technologies[J]. Journal of Experiments in Fluids Mechanics, 2014, 28(2):1-6.

[4] Cong Chenghua, Liao Daxiong, Chen Jiming, Qin Honggang. Numerical investigation on performance of second throat in transonic wind tunnel[J]. Journal of Aerospace Power, 2010, 25(9):2050-2056.

[5] Li Hongzhe, Liao Daxiong, Cong Chenghua. Numerical simulation of flow conditioning device design in wide angle diffuser of continuous transonic wind tunnels[J]. ACTA Aerodynamics Sinica, 2015, 33(2): 198-203.

[6] Cong Chenghua, Ren Zebin, Yang Gaoqiang, Wang Ning. Numerical simulation for optimal design of wide angle diffuser in limited condition[J]. Journal of Aerospace Power, 2016, 31(4): 910-917.

[7] Moonen P, Blocken B, Carmeliet J. Numerical modelling of the flow conditions in a closed-circuit low-speed wind tunnel[J]. Journal of Wind Engineering and Industrial Aerodynamics, 2006, 94:699-723.

[8] Li Qichen, Yang Zhigang. Application of CFD for the desing of aero-acoustic wind tunnel[J]. ACTA Aerodynamics Sinica, 2009, 27(3):373-377.

[9] Dai Yi, Chen Zuogang, Ma Ning, Ren Zebin. Numerical simulation of flow field inside the low-speed wind tunnel[J]. ACTA Aerodynamics Sinica, 2014, 32(2):203-208.