Research & Reviews: Journal of Physics
STM JOURNALS
http://sciencejournals.stmjournals.in/index.php/RRJoPHY/index
ISSN: 2278-2265 (Online)
ISSN: 2347-9973 (Print)
Volume 10, Issue 1, 2021
DOI (Journal): 10.37591/RRJoPHY
Research RRJoPHY

Simulation Studies of a Triangular Shape Closed Loop Wind Tunnel

Kollati Nanda Sai Kumar[1], J. Mahesh[2], Roopsandeep Bammidi[3,*]

Abstract

An innovative approach in the modeling and simulation of flow parameters of a triangular shape closed-loop wind tunnel was carried out using Computational Fluid Dynamics (CFD), which decreases the overall heat loss of the tunnel from the traditional closed loop wind tunnels and individual section simulations are done to predict and prove the decrease in losses occurred by them and the entire concept wind tunnel is simulated to see the uniformity of flow in the test section. The boundary conditions are appropriately selected based on a subsonic closed loop wind tunnel. This wind tunnel is modeled with Guide vane radiator which is used for directing the air for circulating and cooling the circulating air in a closed loop wind tunnel.

Keywords: CFD, guide vane radiator, wind tunnel, closed loop, simulation

INTRODUCTION

A wind tunnel plays a crucial role as an investigation device used in aerodynamic research. The technology in present scenario is in need for experimentation of products like PCB and many other heat involvement applications. And there is always a problem of increase in temperature in a closed loop wind tunnel which can be solved by radiator but by involving a radiator there is always increase in the power consumed by the fan in that wind tunnel. This study provides a solution for all the above problems by decreasing the overall heat loss of the wind tunnel by changing the corner from 90 to 45° with guide vane radiators which serves both the purpose of guide vane and radiator; where the principle components of a wind tunnel [1] include the contraction, test section [2, 3] and diffuser section [4] which are modeled from a base journal and a methodology was developed to predict and prove the individual losses by each part of the wind tunnel using CFD. Various wind tunnels of close loop type have been existed and being developed for applications in aerodynamics and thermal applicative cooling [5]. Recent developments in computational methods have significantly increased the use of numerical models. Numerical methods are often used jointly with physical experimentation such as wind tunnel flow parameters validation and making changes in the model for increasing performance [6]. Several works done previously by various authors for evolution of wind tunnels. Calautit *et al.* validated the importance of guide vanes for upstream and downstream flow in a closed loop wind tunnel [7]. Moonen *et al.* established a methodology for numerically modeling the flow conditions in a closed-circuit low speed wind tunnel system and modifications that are required to achieve a compact and cost-effective wind tunnel design [8].

***Author for Correspondence**
Roopsandeep Bammidi,

[1]UG Student, Department of Mechanical Engineering, Anil Neerukonda Institute of Technology and Sciences (A), Visakhapatnam, Andhra Pradesh, India
[2]Assistant Professor, Department of Mechanical Engineering, Anil Neerukonda Institute of Technology and Sciences (A), Visakhapatnam, Andhra Pradesh, India
[3]Assistant Professor, Department of Mechanical Engineering, Aditya Institute of Technology and Management (A), Tekkali, Srikakulam, Andhra Pradesh, India

Received Date: 11-20-2020
Accepted Date:11-21-2020
Published Date:04-04-2021

Citation: Kollati Nanda Sai Kumar, J. Mahesh, Roopsandeep Bammidi. Simulation Studies of a Triangular Shape Closed Loop Wind Tunnel. Research & Reviews: Journal of Physics. 2021; 10(1): 14–25p.

RESULTS AND DISCUSSION

The research findings proved that vertical guide vanes reduce skewness and angularity, that yields an outstanding quality flow while horizontal guide vanes were initiated by having a slightly decrease in quality of flow.

METHOD, MODELING AND SIMULATION

The simulation is started with air entering the inlet duct which has a cross-section area of 0.75 m×1 m and passes through the 90° corner with guide vanes, and flows through the tunnel through the effuser section of 1 m×1 m area and with a length of 1 m. Here the area of the section goes on decreasing up to 0.5 m×0.5 m to have an increase in velocity by decreasing pressure. From there, the wind enters the test section of 0.5 m×0.5 m, where the body to be tested is placed for experimentation. The air after passing through the test section is followed in the diffuser having inlet area of 0.5 m×0.5 m and outlet area of 0.75 m×1 m with included angles of 4 and 8°. The air enters the 45° corner as shown in Figure 1 and changes its direction. Guide vanes direct the air to a specific direction without compromising on the uniformity. The guide vanes are so modeled that they also have a provision for the flow of coolant through it so that cooling of the air takes place without use of the radiator. Hence the guide vanes can be used both for directing the air and for the cooling purpose. The air is then made to flow in a straight path of 0.75 m×1 m area and length 5.4 m called hypotenuse side to reach another 45° corner. The air again follows the same procedure as done in the previous 45° corner and decreases the temperature of air to a much lower value. Then finally the air is made to flow through the outlet duct of 0.75 m×1 m area. The air is made to circulate in the same manner and cooling is also done simultaneously to get the appropriate conditions suitable for the test specimen. Three sets of 2d simulations were done to compare the major losses in the regular wind tunnel with radiators and wind tunnel with guide vane radiators. This includes the structured and unstructured meshing.

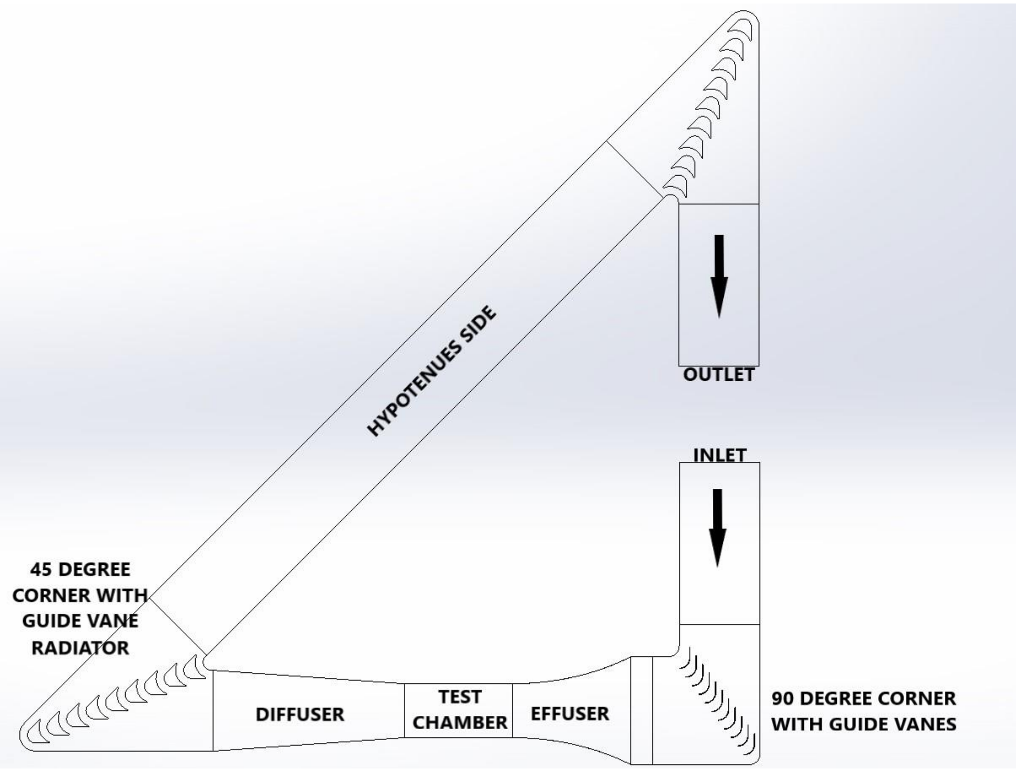

Figure 1. Line diagram.

90° Corner Simulation

The first set of simulation consisting of a structured mesh is done with ICEM CFD software and an unstructured mesh with ANSYS AUTODYN software of a 90° corner as shown in Figure 2, a regular closed loop wind tunnel; in both the meshes, very fine 2d elements are

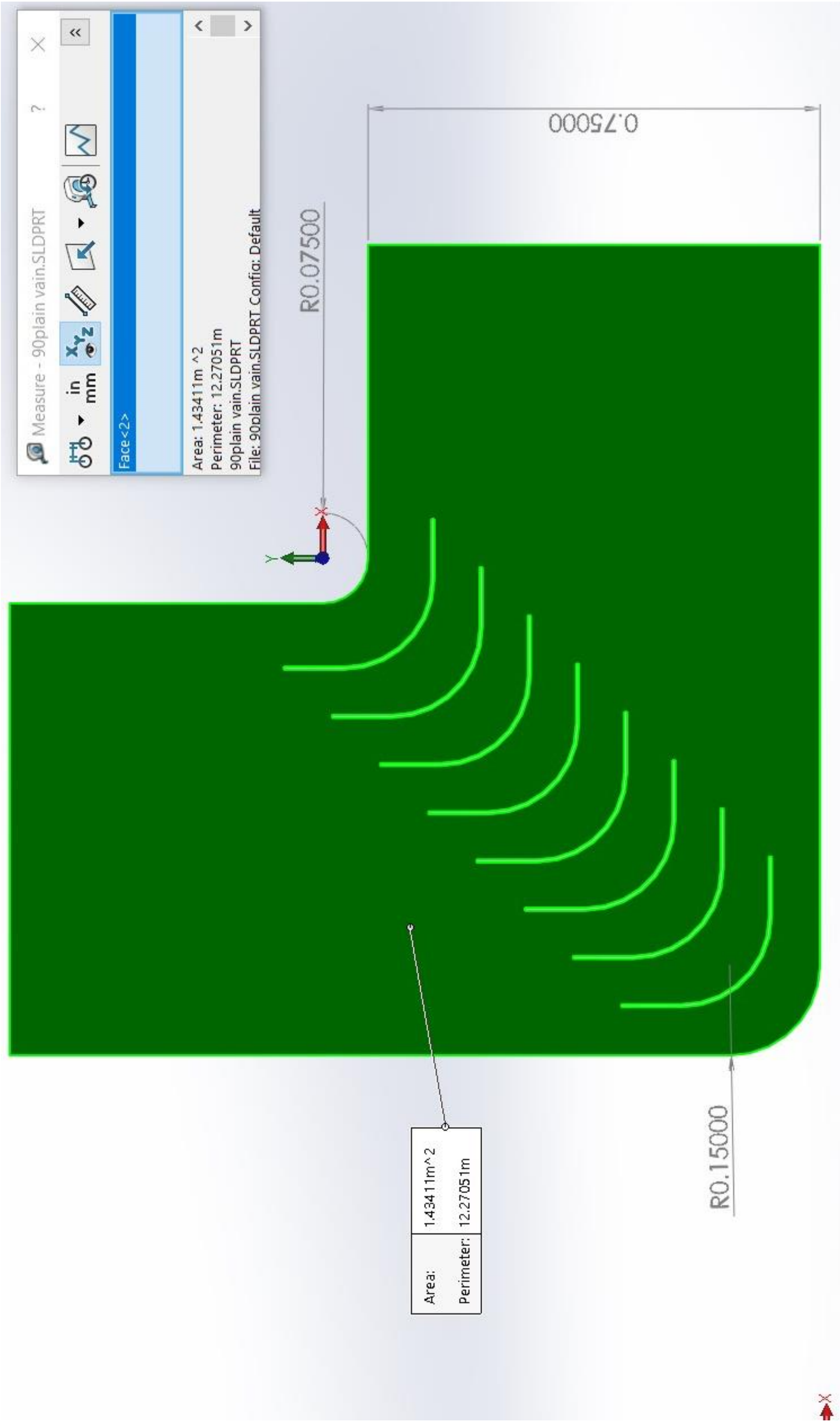

Figure 2. 90° Guide vane model.

used; the dimension of the section are specified below, and for creation of geometry, SOLIDWORKS software is used and later exported to IGS format to import the files to meshing software.

ANSYS FLUENT is used to predict the flow patterns and calculate the loss of that section. The boundary conditions given are summarized below:

From Figures 3 and 4, pressure outlet is monitored using "report definitions" in ANSYS FLUENT for convergence of solution. The results obtained are as follows:

Total pressure at inlet =0.26733 pa, and
Total pressure at outlet =0.22145 pa.

45° Corner Simulation

The second set of simulations consists of same set of conditions as first set but with 45° corner. The boundary conditions used are same that are used in first set and tabulated in Table 1.

Table 1. Boundary conditions.

Parameter	Set Value
Discretisation scheme	Second-order upwind
Algorithm	SIMPLE
Turbulence model	k-epsilon
Time	Steady state
Gravity	–9.81 m/s^2
Inlet	Velocity inlet type with 0.5 m/s
Outlet	Outflow type
Density	Constant (1.225 kg/m^3)
Operating pressure	0 pa

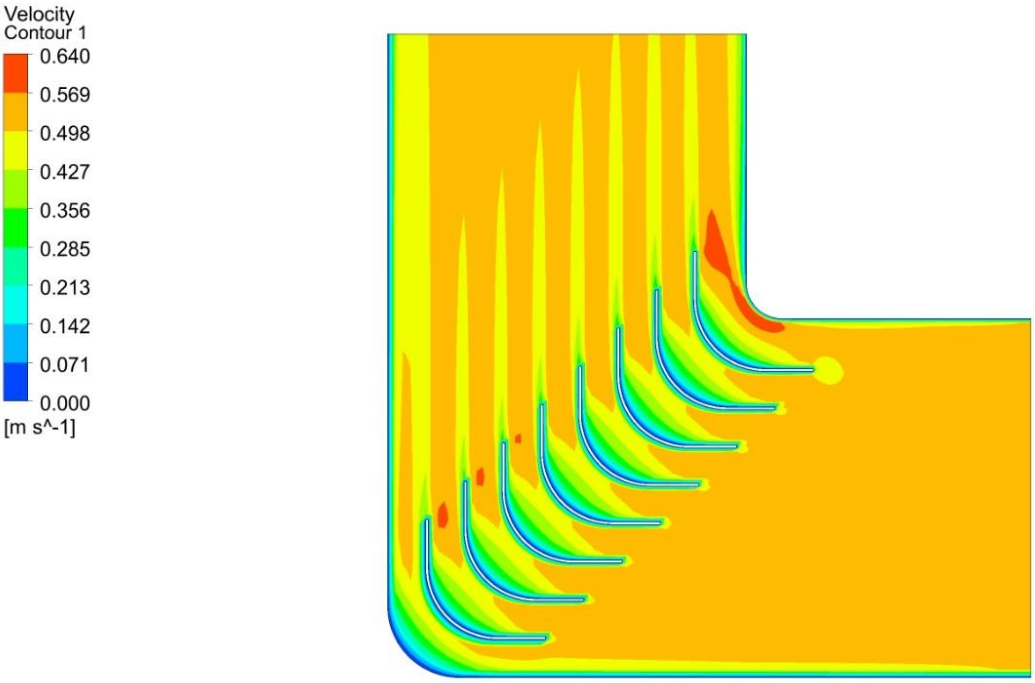

Figure 3. Velocity contoursof 90° model.

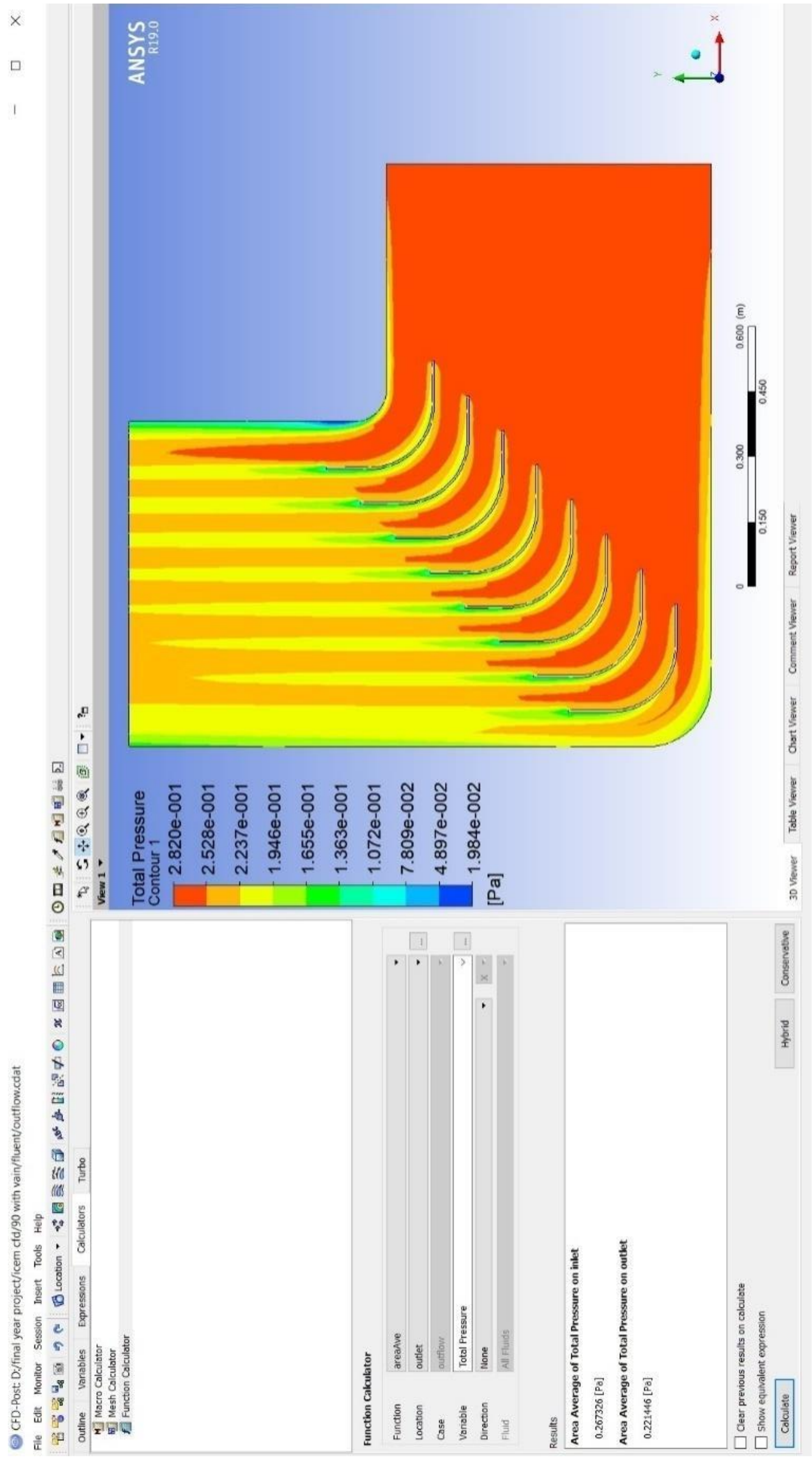

Figure 4. Pressure contours of 90° model.

The results obtained are as follows:

- Total pressure at inlet =0.13623 pa, and
- Total pressure at outlet =0.06052 pa.

From Figures 5–7, the pressure and velocity contours are found.

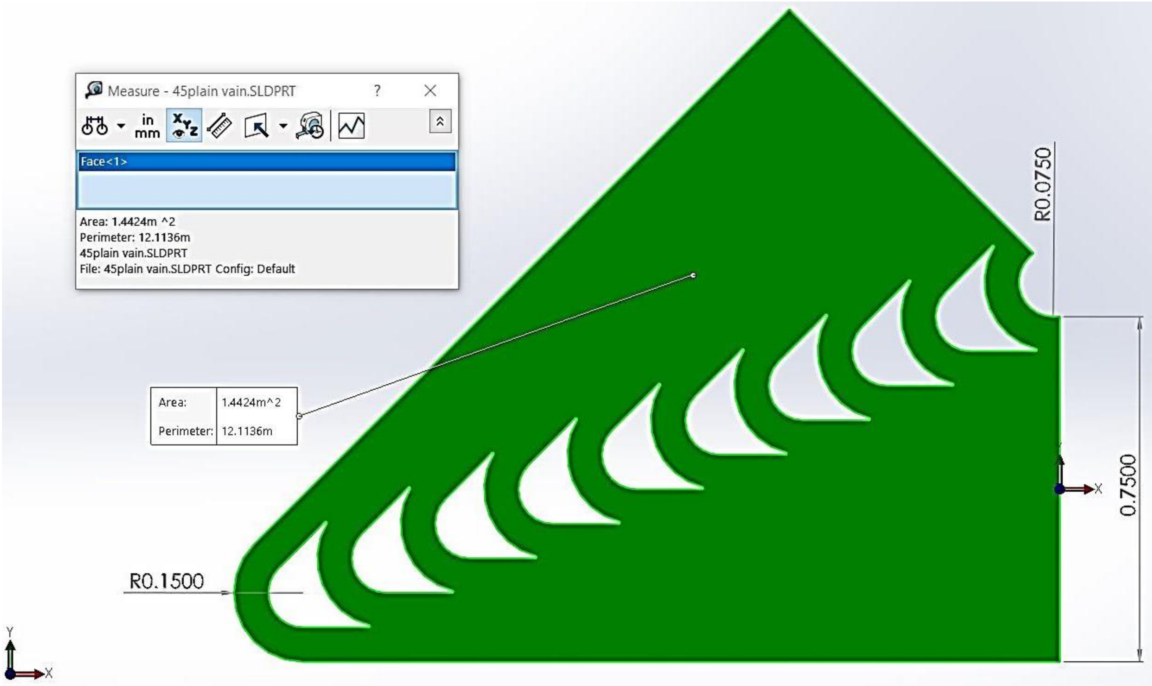

Figure 5. 45° Guide vane radiator model.

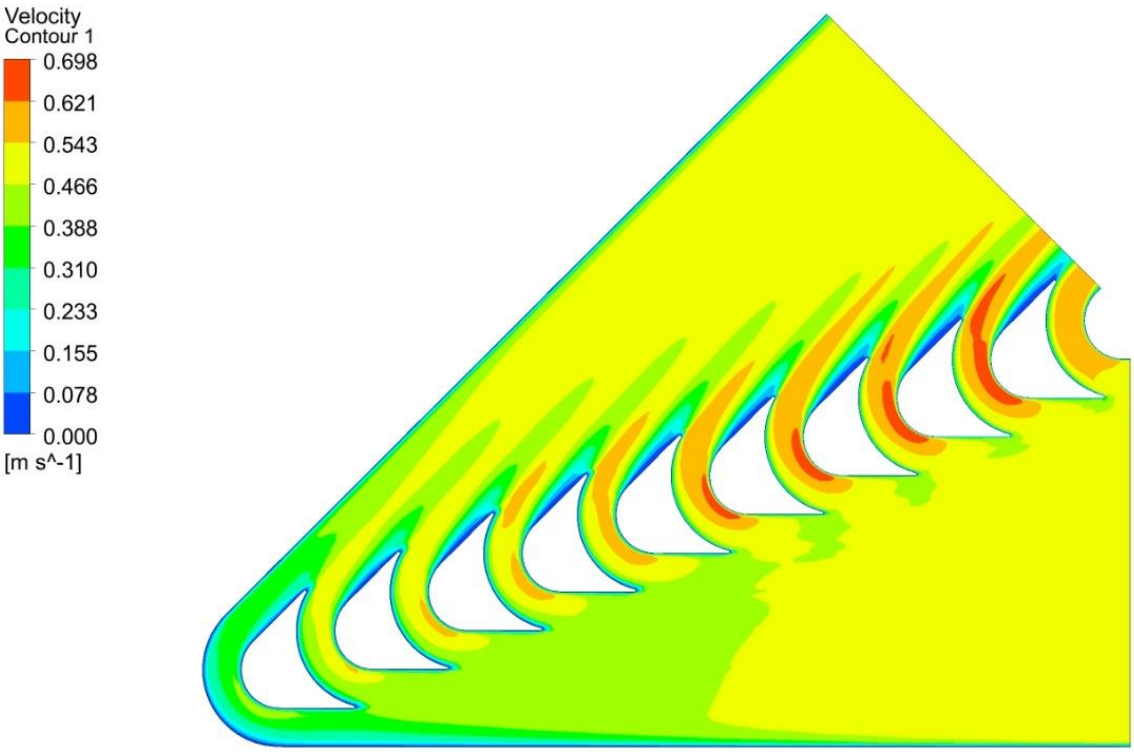

Figure 6. Velocity contours of 90° model.

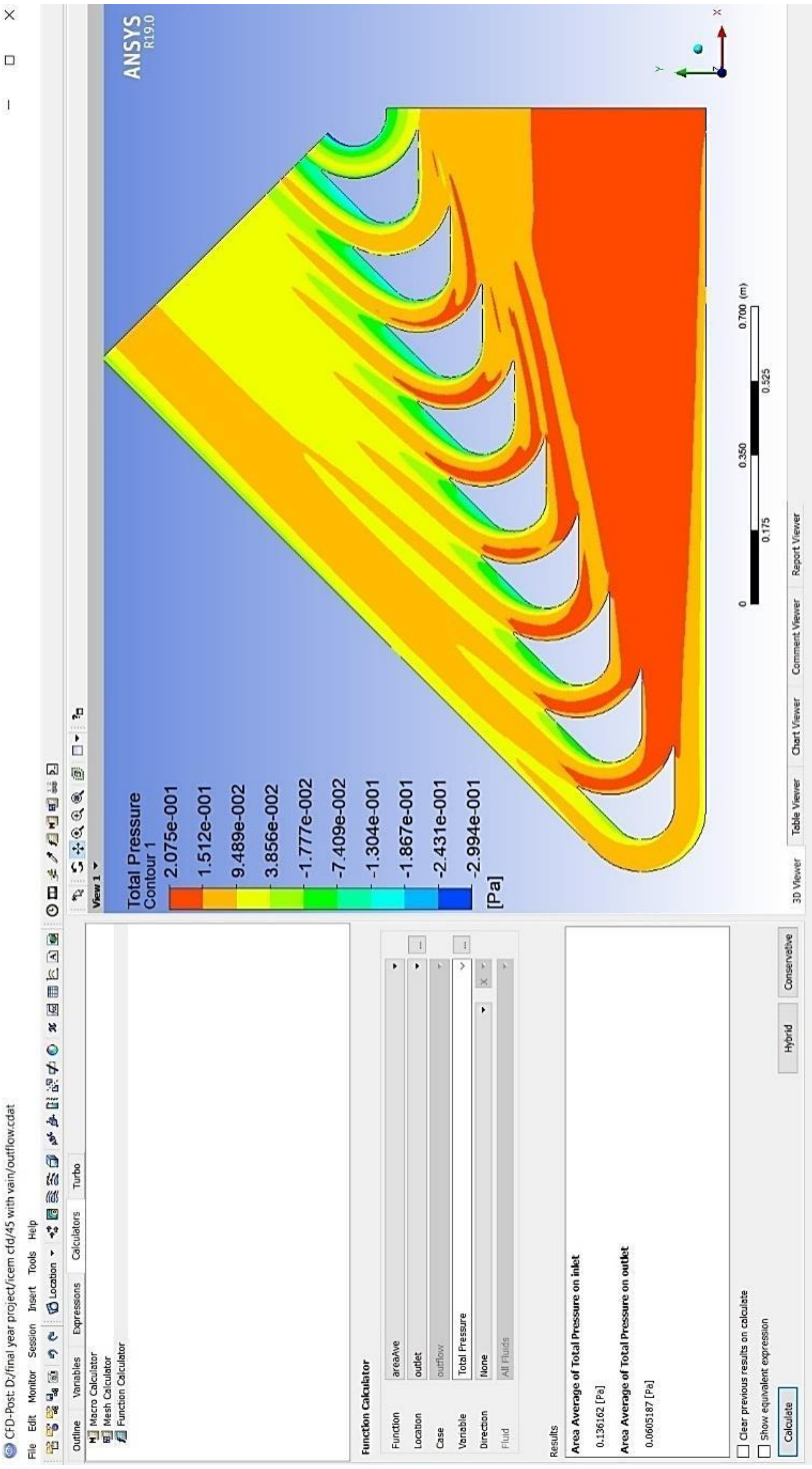

Figure 7. *Pressure contours of 90° model.*

Radiator Simulation

These sets of simulations consist of a rectangular duct with radiators. The duct geometry is identical to the previous 45 and 90° corners. These simulations are done to predict the loss due to a typical radiator that was used in general closed loop wind tunnel. The boundary conditions used are also the same that are used in previous cases tabulated in Table 1.

The results obtained are as follows:

- Total pressure at inlet =0.15082 pa, and
- Total pressure at outlet =0.031396 pa.

From Figures 8–11, all the countours from radiation simulation and required calculations are framed and done.

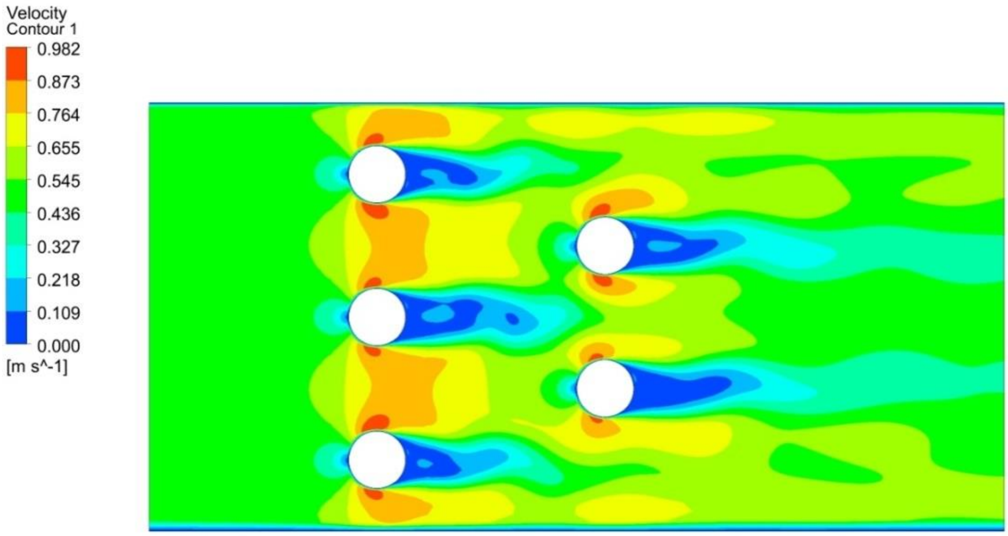

Figure 8. Velocity contour of a traditional radiator section.

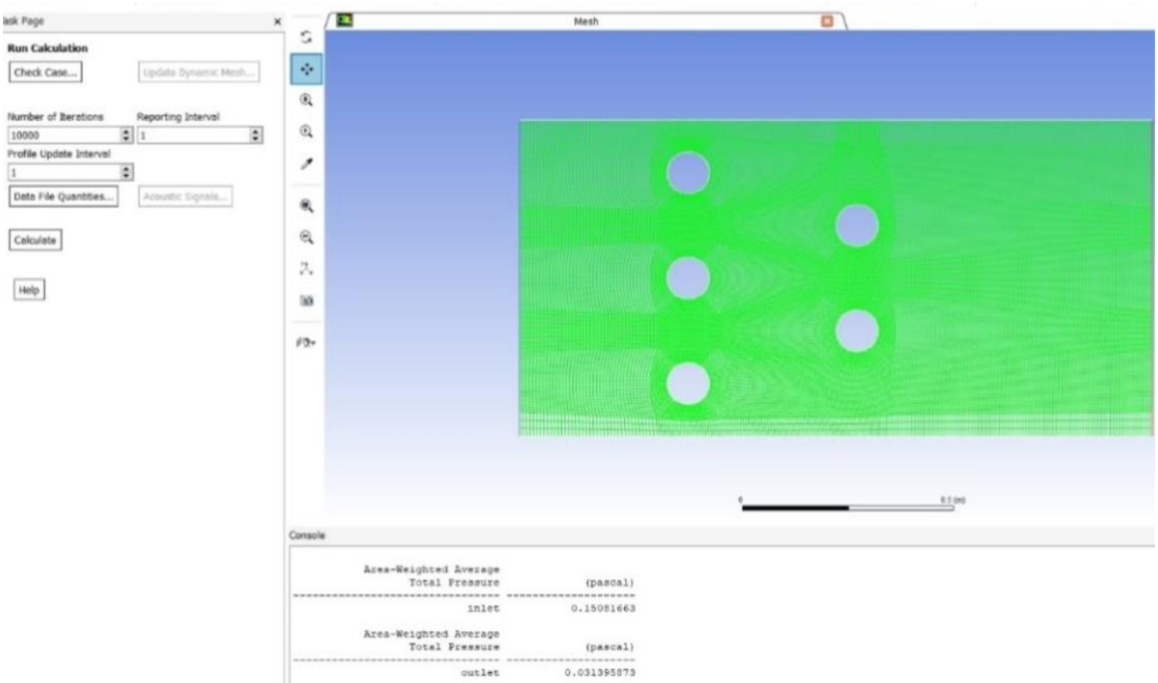

Figure 9. Calculating Total pressures of a traditional radiator section.

Figure 10. Velocity contour of an empty section.

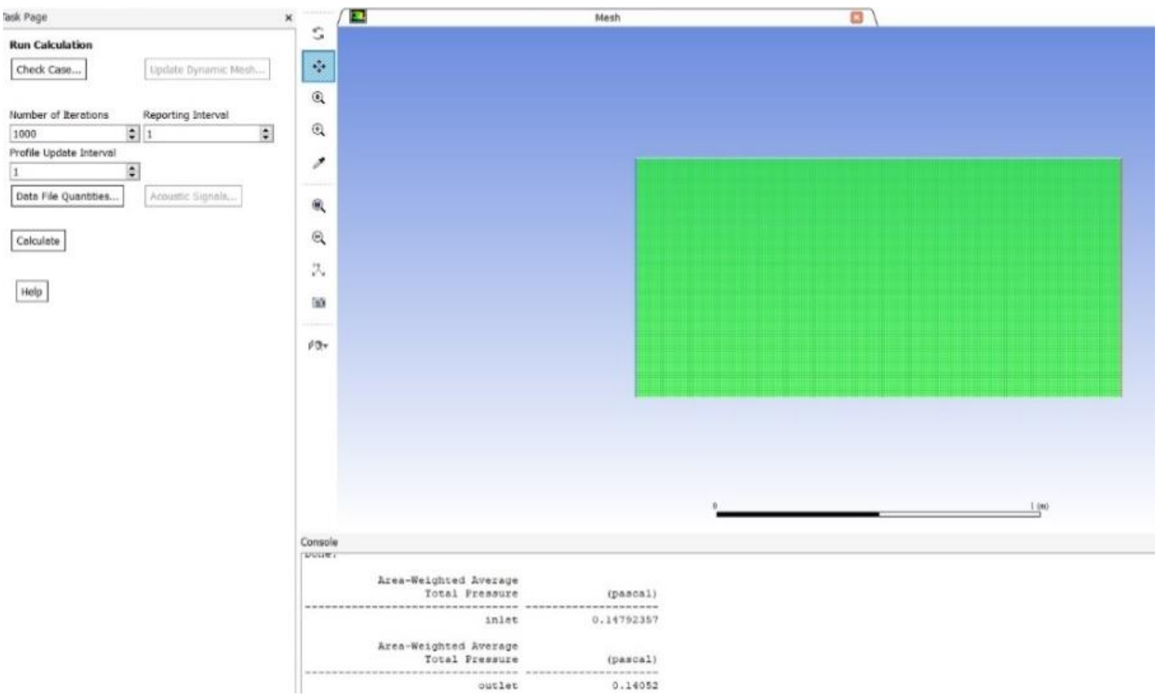

Figure 11. Calculation of total pressuresof an empty section.
Total pressure at inlet =0.14792 pa, and
Total pressure at outlet =0.14052 pa.

loss only due to radiator = *Part 5* = loss (*Part 3 – Part 4*) =*0.0093219*

Now overall major losses in a rectangular closed loop wind tunnel will be four times the loss of *Part 1* and onetime the loss of *Part 5* (if introduced) (Table 2).
=*0.015271+0.0093219*
=*0.024597* m

Overall major loss of conceptual closed loop wind tunnel with guide vane radiator will be two times the loss of *Part 2* and onetime the loss of *Part 1.*
=*0.0126002+0.0038178*
=*0.016418* m.

Hence the overall head loss is 0.008182 m less than traditional closed loop wind tunnel at a velocity of 0.5 m/s and the loss increases as the velocity gradient increases.

3D Wind Tunnel Simulation

The entire concept wind tunnel is simulated to find out the velocity contours, and uniformity in the test section. The boundary conditions are appropriately selected considering the cross section of inlet and test chamber; it was assumed to have 31 m/s which is general velocity of a subsonic closed loop wind tunnel using continuity equation; velocity at inlet is calculated and given as inlet boundary condition. For grid independence study, the mesh is varied up to nodes of 1077965 and elements of 5904875. The minimum size of the 0.00049275 m, proximity and curvature with a growth rate of 1.2 minimum qualities is of 0.17 and all the elements are tetrahedral. The residuals are reduced for accurate results, second order upwind is used for all turbulence models starting with k-epsilon and then changed to k-e Renormalization Group (RNG) turbulence model, the k-w Standard Shear-Stress Transport (SST) model and the Reynolds-Stress Model (RSM) with Linear Pressure-Strain and Stress-Omega models are compared later on. The contours obtained are as follows (Figures 12–15):

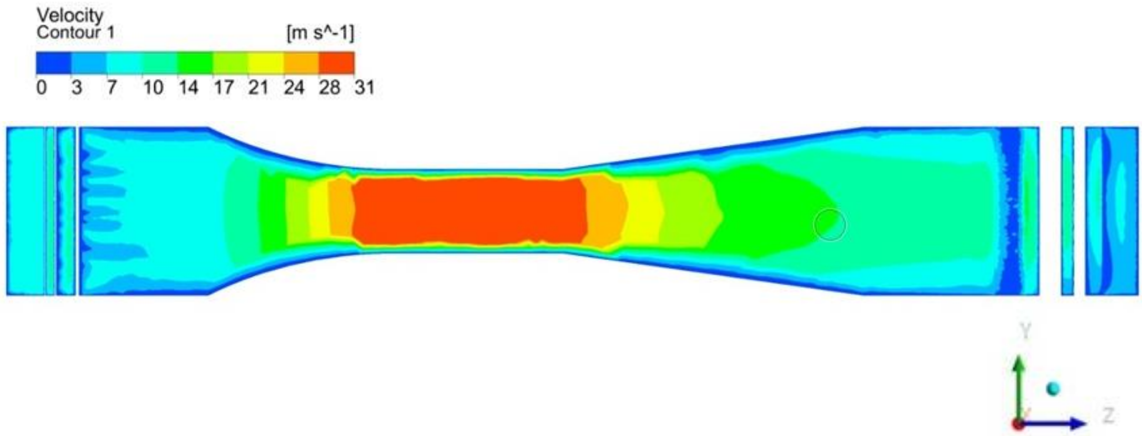

Figure 12. Front view of the wind tunnel.

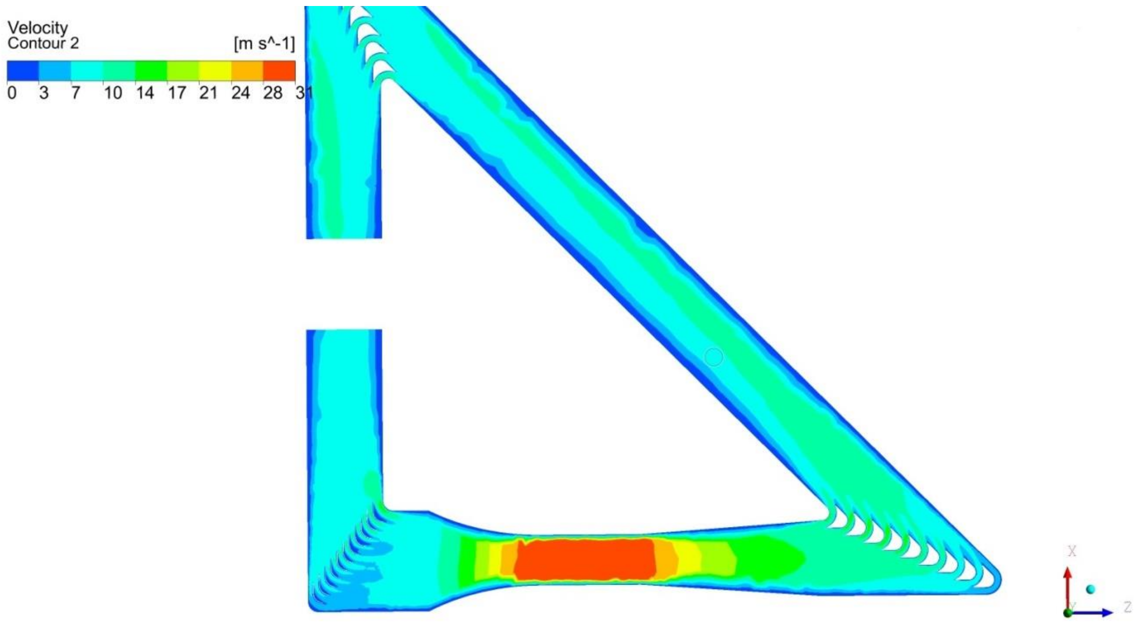

Figure 13. Top view of the wind tunnel.

ISSN: 2278-2265 (Online), ISSN: 2347-9973 (Print)

Table 2. Obtained head losses through simulations.

Name	Description	Total Pressure at Inlet (P_1) (in bar)	Total Pressure at Outlet (P_2) (in bar)	Head Loss = $(P_1–P_2)/\rho*g$ (in bar)
Part 1	The 90° corner	0.26733	0.22145	0.0038178
Part 2	The 45° corner	0.13623	0.06052	0.0063001
Part 3	The duct with radiator	0.15082	0.031396	0.0099377
Part 4	The duct without radiator	0.14792	0.14052	0.0006158

Summary of Velocity Flow Distributions in the Test Section

The power required to maintain a steady flow in the wind tunnel is equal to the total losses occurring in each section or the flow throughout the wind tunnel that appears as a decrease in total

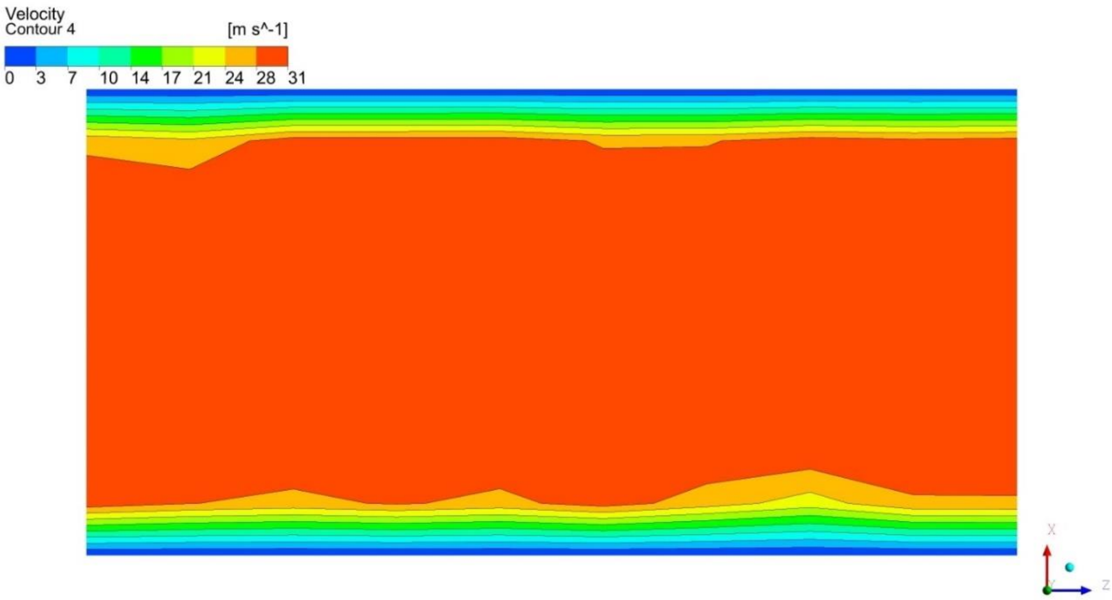

Figure 14. Front view of the test section.

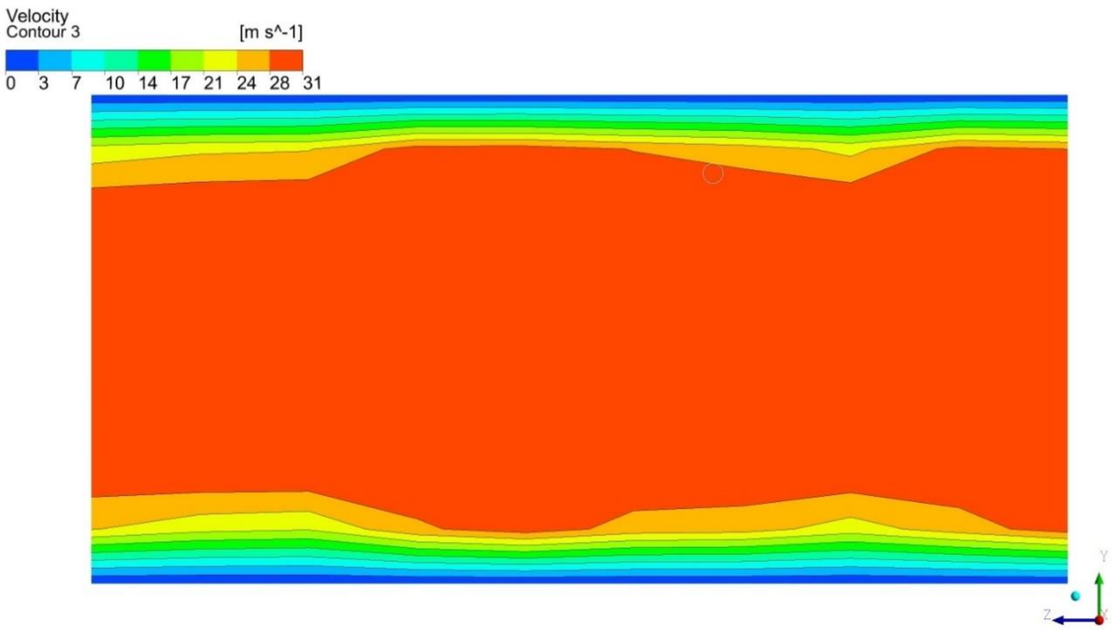

Figure 15. Top view of the test section.

pressure, must be compensated by a pressure rises as well as loss in kinetic energy; in this case provided by the fan. The total pressure that should be provided by the fan is calculated using CFD simulation is of 119.80 pa.

CONCLUSION

A numerical investigation is carried out into modeling simulation of flow parameters in closed-loop subsonic wind tunnel by incorporating Computational Fluid Dynamics. The power required to the concept wind tunnel is less compared to a traditional wind tunnel with radiator and that is proved using 2D simulations of individual parts which plays a major role in resistances by creating loss that should be overcome by the fan. It also provides a solution for the problem of heat generation in a closed loop wind tunnel by incorporating the radiator in the guide vanes in an efficient manner, which is named as guide vane radiator. This study provides an idea of varying the temperature of air in the closed loop wind tunnel for reaching the experimental needs of modern industries.

REFERENCES

1. Wittwer Adrian R, Moller Sergio V. Characteristics of the Low Speed Wind Tunnel of the UNNE. *J Wind Eng Ind Aerodyn.* 2000; 84(3): 307–320p.
2. Morimasa Watakabe, Masamiki Ohashi. Comparison of Wind Pressure Measurements on Tower-Like Structure Obtained from Full-Scale Observation, Wind Tunnel Test, and the CFD Technology. *J Wind Eng Ind Aerodyn.* 2002; 90(12–15): 1817–1829p.
3. Sahin B, Ward-Smith AJ. The Pressure Drop and Flow Characteristics of Wide Angle Screened Diffusers of Large Area Ratio. *J Wind Eng Ind Aerodyn.* 1995; 58(1–2): 33–50p.
4. Kevin Owen F, Owen Andrew K. Measurement and Assessment of Wind Tunnel Flow Quality. *Prog Aerosp Sci.* 2008; 44(5): 315–348p.
5. John Kaiser Calautit, Hassam Nasarullah Chaudhry. Comparison between Evaporative Cooling and a Heat Pipe Assisted Thermal Loop for a Commercial Wind Tower in Hot and Dry Climatic Conditions. *Appl Energy*. 2013; 101: 740–755p.
6. Peter Moonen, Bert Blocken. Indicators for the Evaluation of Wind Tunnel Test Section Flow Quality and Application to a Numerical Closed-Circuit Wind Tunnel. *J Wind Eng Ind Aerodyn.* 2007; 95: 1289–1314p.
7. John Kaiser Calautit, Hassam Nasarullah Chaudhry. A Validated Design Methodology for a Closed-Loop Subsonic Wind Tunnel. *J Wind Eng Ind Aerodyn.* 2014; 125: 180–194p.
8. Peter Moonen, Bert Blocken. Numerical Modeling of the Flow Conditions in a Closed-Circuit Low-Speed Wind Tunnel. *J Wind Eng Ind Aerodyn.* 2006; 94(10): 699–723p.

