NUMERICAL ANALYSIS OF VELOCITY VECTORS PLOTS AND TURBULENT KINETIC ENERGY PLOTS OF FLOW OF THE AIR CURTAIN

Mr Nitin Kardekar, Principal, Jayawantrao Sawant Polytechnic.
Research Scholar, Singhania University.
Dr. V K Bhojwani, Professor, JSPM’s Jayawantrao Sawant College of Engineering, Pune
Dr Sane N K, Research Supervisor, Singhania University

ABSTRACT

A prototype is developed in the laboratory in order to simulate the conditions of the entrance of the doorway installed with air curtain device. The air curtain blows the air in downward direction. The flow within the air curtain is simulated with commercial Computational Fluid Dynamics (CFD) solver, where the momentum equation is modelled with Reynolds-Average Navier-Stokes (RANS)’ K- ε turbulence model. The boundary conditions are set up similar to the experimental conditions. The CFD results are compared and validated against experimental results. The results are obtained in the form of contours; which are plotted for velocity and turbulent kinetic energy. The velocity vectors are studied to observe the infiltration through the air curtain and weak velocity zones. High turbulent zones are identified using the turbulent kinetic energy plots.

Key words: Air curtain, Reynolds-averaged Navier – Stokes equation, K- ε turbulence model, Velocity vectors, turbulent kinetic energy.

INTRODUCTION

Air curtain devices provide a dynamic barrier instead of physical barrier between two adjoining areas (conditioned and unconditioned) thereby allowing physical access between them. The air curtain consist of fan unit that produces the air jet forming barrier to heat, moisture, dust, odours, insects etc. The Air curtains are extensively used in cold rooms, display cabinets, entrance of retail store, banks and similar frequently used entrances. Study found that air curtains are also finding applications in avoiding smoke propagation, biological controls and explosive detection portals. According to research by US department of energy
1875 MW energy will be saved per year by optimising the performance of super market display cabinet air curtains. In 2002 the UK food and drinks industry used equivalent of 285 tonnes of oil to power its refrigeration units with most being used in cold storages. In developing countries like India; the rise in cold storages, super markets, retail stores, banks are not only limited to mega cities but they have also become an integral part of suburban’s and small towns. The effects of globalisation are inevitable. The air curtains are no more luxury but are necessary part of business development and economy. Hence study of air curtain with respect to Indian climate is necessary to ensure optimised performance of air curtains which would leads to energy conservation. The saving of energy (Electrical energy) will be always boon for energy starving country like India.

METHODOLOGY

The air flow analysis was carried out using commercial software package ANSYS V13.0 Workbench platform. As shown in figure 1 the air curtain is mounted on the top of the frame. The doorway frame chosen is 2270 mm in height and 900 mm in width, the breadth of the frame is 290 mm. There are two slits which open in the air flow domain; the flow jet is pushed by the blower in the domain through these two slits which are 84 mm apart. (Refer figure 1) The entire experiment is carried out at isothermal conditions; ambient air at 29\(^{\circ}\)C (+- 1\(^{\circ}\)C) at one atmosphere. The velocity of air leaving the slits is measured 7.6 m/s. All these conditions are used for CFD analysis. This velocity is representative of air curtain flow velocity. The flow domain is extended to capture the flow leaving frame boundaries in directions of frame openings. The frame walls are treated as impermeable walls, and are ‘no slip’ walls. It is ensured while choosing the length of extended domain that the direct transverse flow of air curtain will not cross the boundaries of the domain. Once the configuration is modelled, the mesh is generated in the workbench. The structured mesh (hexahedron mesh) is used to build the extended domain and flow straightener. The frame portion is meshed with unstructured tetra mesh.

![Figure 1 Prototype of model](image1.png)

![Figure 2 Geometry model](image2.png)
The effort was made to mesh the entire domain with structured mesh but due to complex geometry at the flow straightener, the frame portion has unstructured mesh. The total mesh count is 385443, within which 59589 are tetrahedral cells and 325854 hexahedral cells. The minimum mesh quality is 0.3, total 708 cells falls within this range, as per the CFD Practices this is a good quality mesh. The mesh which is created in the Workbench is internally transferred to CFX-Pre, a CFD solver available with workbench platform. The flow within the air curtain is simulated within commercial Computational Fluid Dynamics (CFD) solver, where the momentum equation is modelled using Reynolds-Average Navier-Stokes (RANS)’ K-ε turbulence model. The default domain is air at 29°C. The inlet boundary condition used is ‘normal speed’ at 7.6 m/s, since the actual turbulence data at inlet is currently unavailable, for the present simulation the uniform turbulence intensity of 5% (medium intensity) is used to model the inlet turbulence. The outlet condition is assigned to the extended domain walls as average static pressure of zero gauge.

The computational platform is HP- Pavilion dv6, with Intel CORE i3 2.4GHz processor, 8GB of RAM. The convergence target is set at 1e-4 RMS; with continuity target error of 1e-4 kg/s. The convergence target is achieved after 160 iterations.
RESULTS AND DISCUSSIONS

The objective of the air curtain is to restrict the infiltration of outside air and thus save the energy and keep the comfort conditions inside the space as it is. The addition of air curtain which serves as an interface/partition between two spaces, it is very complicated flow pattern observed in close vicinity of the air curtain device. The analysis of such a complicated flow pattern observed in close vicinity of the air curtain device.
is difficult with experiments, whereas CFD can be a useful tool to analyze the flow patterns when validated. The velocity matrix generated with experiments at the center plane is in good agreement with the CFD results. The figure 5 shows the result comparison. The validation result justifies the CFD results, and thus extends our belief for other flow results generated which are impossible to judge experimentally. The CFD plots associated with 3 mid planes are as shown in the figure 6. Figure 7 shows transverse velocity contour on plane 3 positioned at right angle to air curtain passing through the slit. It reveals that the flow velocity of air curtain reduces to 2.3 m/s from initial velocity 7.6 m/s till it reaches ground and diverts sideways. The figure 9 shows velocity vector at the same plane which indicates that the velocity vectors from both sides of the air curtain are not crossing the barrier. This satisfies the condition of separation of environment on the either side of the curtain. The cross flow velocities are found in the range of 0 to 1 m/s. It is clear from figure 7 that cross velocity vectors on plane 1 are attracted towards low pressure zone of air jet.
then directed downside because of high momentum of air jet which assures non cross flow conditions. But, when cross flow velocity vectors were observed at plane 2 passing through the space between the slits, revealed that the velocity vectors of the magnitude 0.1 to 0.83 m/s are passing through small dead portion between 2 slits (Below motor support base). This is the portion where air curtain is weak barrier. Figure 8 also supports this finding, that the flow in this region is not a downward barrier flow but, it is diffused flow due to the geometry of the air curtain. This may be the reason why air curtain is breached marginally. Hence it is concluded to avoid separation of air entry slits if possible and if not from manufacturing point of view, then should be kept minimal to improve the barrier effectiveness.

Figure 10 shows that turbulence kinetic energy contour at YZ plane across the air curtain. It is observed that initially turbulence is less and as it approaches at the ground the turbulent kinetic energy (TKE) increases, the range of TKE is of 0.39 m²/s² to 0.43 m²/s². This represents smooth air curtain flow initially and disturbances occur in the flow when it reaches the ground. When jet of air curtain strikes the ground it is diverted sideways, which causes higher TKE. When kinetic turbulence energy plots were observed from front maximum TKE was observed near ground. However when observed in the plane 2 (figure 11) the higher turbulence kinetic energy is observed between height of 200 mm to 800 mm from the top. The air curtain expands in XZ plane may be the reason of higher values of TKE. Figure 12 shows the velocity streamlines starting at inlet which are smooth flow line reaching the ground and diverting on either sides. The flow of air is found continuous, straight and without break. This is the desired function of air curtain. In many applications especially in refrigerated air curtains the need is to divert the flow only on one side (Inside conditioned space). This can be accomplished by varying the jet angle with the help of guide vane in the air curtain.

CONCLUSION

A numerical study of air curtain flow over door way was performed using CFD code Ansys CFX 13.0. The study found the model is in good agreement with the experimental results. The flow over curtain was found continuous, straight and without break, as per requirement of the air curtain. The plot of turbulent kinetic energy shows the higher turbulent region of the flow of the air curtain. The cross flow is the cause due to dead zone and not because of flow behaviour. The region is the area of air curtain where flow is blocked because of the base of air curtain motor. The velocity vectors of magnitude 0-0.83 m/s are found crossing air curtain. It is recommended to avoid separation of air entry slits if possible and if not from manufacturing point of view, then should be kept minimal.

REFERENCE


