

Enhancement Of Fire Evacuation System In Car Parking Area Using CFD

B Karthik¹

¹(Mechanical Engineering, Hindusthan College of Engineering & Technology, Coimbatore, India, karthik.race007@gmail.com)

Abstract— When a fire breaks out in a building, a large amount of smoke is generated at high temperature leading to poor visibility and difficulty in evacuation. In most cases, the victims are poisoned or starved of oxygen by the smoke. In addition, the smoke also causes damage to neighboring buildings. We use FDS (Fire Dynamics Simulator) model for designing and checking the effectiveness of the smoke ventilation system. The applications of this model are used extensively in car parks, atriums, tunnels, hospitals, airports and shopping complex etc.

Keywords— Jetfans, smoke simulator, smoke sensor, exhaust system

1. INTRODUCTION

In commercial car parking area, when fire breaks out it is very difficult to stop the fire with normal preventive techniques. To overcome this situation we use MEP (Mechanical Electrical Plumbing) & fire safety design. In this jet fans operating at high speed to stop the fire and to find the most suitable place to place the device to achieve maximum efficiency. Also to remove the propagated smoke (due to fire) by using exhaust fans & provide adequate ventilation. Underground parking garages come in a variety of design characteristics. They can be fully or partially closed, partially or completely underground, consist of one floor or several and vary in motor capacity, usage, congestion and size. While only some of the partially closed garages are required by standard and regulations to install an electro-mechanical Smoke Control and Ventilation (SCV) systems, fully enclosed garages that exceed one underground level are always required to install such systems. Underground garages are unique in their low clearance ceiling height between 2.2m to 3m (including cable and duct area) and by being a large compartment area that usually lacks separation by walls and distinct fire zones. In addition, the use of fire resistant walls and doors is rarely present.

The main objectives of this project are as follows: During fire accident situations its difficult to clear the propagated smoke. In our project we are using jet fans to clear the smoke.

- To minimize the rate of fire as soon as possible and to minimize ASET (Available Safe Egress Time)
- To maximize RSET (Required Safe Egress Time)
- To minimize the fire propagated at minimum time.
- To minimize smoke and ensure ventilation quickly.
- To ensure the people can safely escape from the building in case of accidents.

The objective of the computational fluid dynamics (CFD) analysis is to examine the effectiveness of designed emergency ventilation system in case of accidental car fire in the basement car parking area. The available safe egress

time (ASET) is the amount of time that elapses between fire ignition and the development of untenable conditions. The required safe egress time (RSET) can be defined as the time period from the ignition of fire until the time the last occupant reaches a place of safety.

2. COMPUTATIONAL FLUID DYNAMICS

Fluid dynamics is a field of science which studies the physical laws governing the flow of fluids under various conditions. Great effort has gone into understanding the governing loss and the nature of fluids themselves, resulting in a complex yet theoretically strong field of research. Computational Fluid Dynamics or CFD as it is popularly known is used to generate flow simulations with the help of computer. CFD involves the solution of the governing loss of fluid dynamics numerically. The complex sets of partial differential equation of solved on in geometrical domain divided into small volumes, commonly known as a mesh (or grid).

Computational fluid dynamics (CFD) is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations which govern these processes using a numerical process. Computers have been used to solve fluid flow problems for many years. Numerous programs have been written to solve either specific problems, or specific classes of problems. From the mid-1970's, the complex mathematics required to generalize the algorithms began to be understood, and general purpose CFD solvers were developed. These began to appear in the early 1980's and required what were then very powerful computers, as well as an in-depth knowledge of fluid dynamics, and large amounts of time to set up simulations. Consequently, CFD was a tool used almost exclusively in research. Recent advances in computing power, together with powerful graphics and interactive 3D manipulation of models, have made the process of creating a CFD model and analyzing results much less labour intensive, reducing time and, hence, cost. As a result of these factors, Computational Fluid Dynamics is now an established industrial design tool, helping to reduce design time scales and improve processes throughout the engineering world. CFD provides a

cost-effective and accurate alternative to scale model testing, with variations on the simulation being performed quickly, offering obvious advantages.

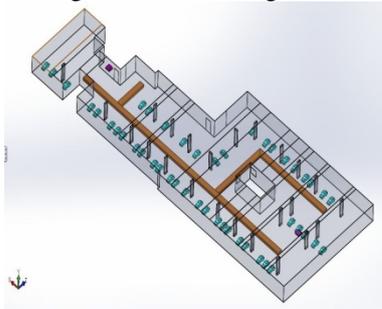


Figure1. solidwork model of car parking area

In many simulations there is need to have fluid flow out of one or more boundaries of the computational region. In compressible flow, when the flow speed at the outflow boundary is supersonic, it makes little difference how the boundary conditions are specified since flow disturbances cannot propagate upstream. In low speed and incompressible flows, however, disturbances introduced at an outflow boundary can have an effect on the entire computational region. The simplest and most commonly used outflow condition is that of “continutive “boundary. Continutive boundary conditions consist of zero normal derivates at the boundary for all quantities. The zero-derivative condition is intended to represent a smooth continuation of the flow through the boundary. As a general rule, a physically meaningful boundary condition. Such as a specified pressure condition, should be used at out flow boundaries, whenever possible. When a continutive condition is used it should be placed as far from the main flow region, as is practical so that any influence on the main flow will be minimal.

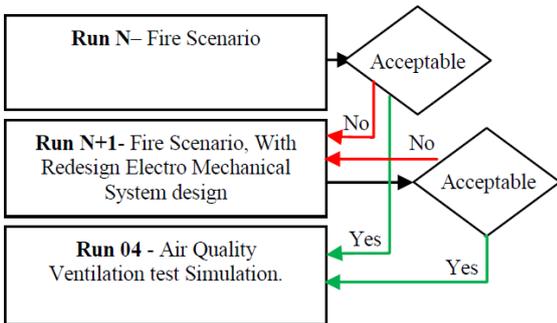


Figure2.design process of simulation

Geometric model is generated in ‘SOLIDWORKS’ which is very popular modeling software. The generated model is exported to the further process in the form of .IGES as it is a third party format which can be taken in to any other tools. Extracting the fluid region is the next step in which all the surfaces which are in the contact of fluid are taken alone and all other surfaces are removed completely. To keep the domain air /water tight some extra surfaces are created. This clean up is done in ANSA meshing tool which is very robust clean

up tool. Extracted domain for vortex generation and finder assemblies are shown below.

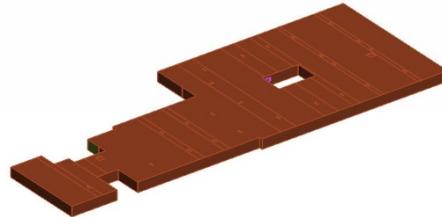


Figure3.Fluid domain extraction of CAD model

3. MESHING

After cleaning up the geometry surface mesh is generated in ANSA tool itself. All the surfaces are discredited using tri surface element .As the geometry has some complicated and skewed surfaces tri surface elements are used to capture the geometry. The following figure shows the surface meshes.

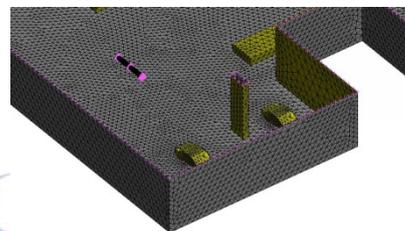


Figure.4. surface mesh on fluid domain

Volume mesh is generated in T-Grid which is a robust volume mesh generator. Volume is dicretized using tetrahedron .Each and every cell centroid is the co-ordinate at which the navier-stokes system of equations are solved.

MESH	COUNT	QUALITY
SURFACE MESH	310696	0.6
VOLUME MESH	1364086	0.83

Figure.5.mesh details

4. SIMULATION RESULTS AND ANALYSIS

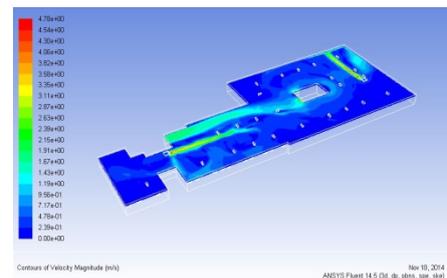


Figure.6.velocity contour

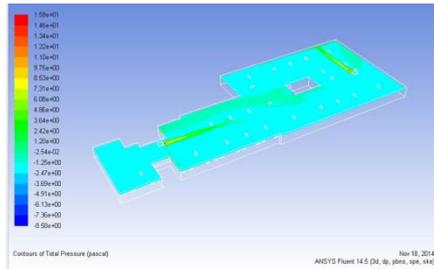


Figure.7.total pressure

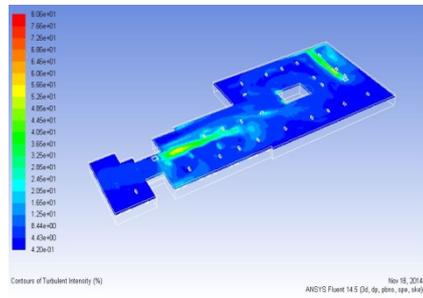


Figure.8.turbulent intensity

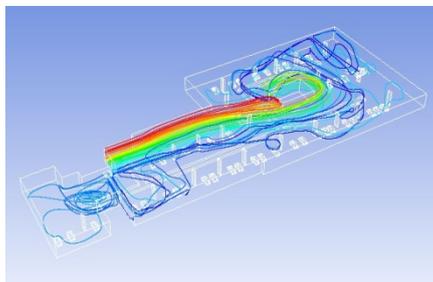


Figure.9.path lines

5. CONCLUSION

Thus a novel approach is taken to analyze and understand the flow behavior in the parking ventilation area. The results show that the velocity is insufficient to drive the flame. In the future study jet fans are to be included in the domain to increase the localised velocity. The appropriate position and numbers can be found through CFD analysis.

Positioning of jet fans and exhaust fans are determined and analysis are taken on the future work. Increasing the flow behaviour and velocity are the targets in our further proceeds.

Advantages

- Increase in safety
- Enhanced ventilation

Applications

- Car parking

REFERENCES

[1] Ning, X., and Lovell, M. R., 2002, "fire and explosion in large car parks," ASME J. Tribol., 124(1), pp. 5-13.

[2] Kwon, O. K., and Pletcher, R. H., 1981, "Prediction of the fire flow in closed car park," Technical Report No. HTL-26, CFD-4, Iowa State Univ., Ames, IA.

[3] Smith, R., 2002, "fire risk analysis on residential building," Ph.D. thesis, <http://www.cas.phys.unm.edu/rsmith/homepage.html>

[4] Tung, C. Y., 1982, "Underground parking garages-changing perceptions on smoke control criteria and combining an integrated system for smoke control and ventilation," Ph.D. thesis, Rensselaer Polytechnic Institute, Troy, NY.

[5] Hashish, M., 2000, "building safety and human behavior in fire," High Pressure Technology, PVP-Vol. 406, pp. 135-140.

[6] Lee, Y., Korpela, S. A., and Horne, R. N., 1982, "car parking fire safety," Proc. 7th International Heat Transfer Conference, U. Grigul et al., eds., Hemisphere, Washington, DC, 2, pp. 221-226.

[7] Jones, J., 2000, fire safety, Cambridge University Press, Cambridge, UK, Chap. 6.

[8] Barnes, M., 2001, "pressure in closed area," J. Appl. Phys., 48(5), pp. 2000-2008

[9] Rajendran I., and Vijayarangan S., Finite element modelling and stress analysis of jet fans, Proceedings of International Conference on CAD/CAM, Robotics and Factories of the Future, The University of the West Indies, St. Augustine -Trinidad, West Indies, 2000, 859-866.

[10] Rajendran I., and Vijayarangan S., Optimal design of car parking safety, 2001, 79(2), 1121-1129.

[11] Dr. Tarek Beji "Research Findings on Ventilation in Tunnels & car parks", IEEE Trans. Ind. Appl., vol. 41, no. 4, pp. 1047-1055, 2005

[12] Dr. Margrethekobes "Building safety and human behavior in fire". in Proc. Int. Conf. Advances in Recent Technologies in fire safety systems, 2009, pp. 456-460.

[13] T.W. Ching, "jet fans and safety," J. Asian safety purpose, vol. 4, no. 2, pp. 911-917, 2006, (2006).

[14] Uzair Ansari, Saqib Alam, Syed Minhaj un Nabi Jafri, "Modeling of jet fans", 2011 UKSim 13th International Conference on Modelling and Simulation, pp. 189-194.

[15] V. U, S. Pola, and K. P. Vittal, "ventilation system and design," in Proc. IEEE Region 10 Conference, 2008, pp. 1-6.