International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 10 Issue: 06 | Jun 2023 www.irjet.net p-ISSN: 2395-0072

CFD modelling calculation and simulation of bus

Sagar Verma[1], Dr. Rahul joshi[2], Dr. Mayank Ladha[3], Mr. Manoj sharma[4],

[1]*ME Student, Swami Vivekanand College of Engineering*
[2]Associate Professor, *Swami Vivekanand College of Engineering*
[3] Associate Professor, *Swami Vivekanand College of Engineering*
[4]Assistant Professor, *Swami Vivekanand College of Engineering*

---***---

Abstract - *The study of his project is to be concentrated on CFD analysis of bus interior for optimization of thermal comfort for the entire passenger in a bus that is available in market. For the upcoming market the need of greener city transportation is on the peak the busses will play as the life line of transportation and main challenges of buses is to optimize the design for the minimum consumption of power and maintain safety and comfort for the passenger. This study will works in this direction the aim of the study is to formulate CFD analysis using CFD tool to optimize the passenger comfort in the bus the main objective to work within a given specifications of dimensions, electrical power requirement of bus AC, and other system to be fulfilled under norms issued by authorities and doing changes to maximize the effectiveness of AC for individual passengers.*

Key Words: CFD, Fluent, polyhedral mesh

1. INTRODUCTION

Computational fluid dynamics (CFD) is a technique of fluid mechanics that uses mathematical numerical analysis and applied data with its physical properties such as pressure, velocity, temperature, density and viscosity to solve and analyze problems that involve fluid flows.

1.1 CFD simulation

The city public transportation bus has the purpose of transporting numerous people and is considered a sustainable means of transportation. The problems that are faced by the electrical vehicles are the overheating of passenger cabin due to lack of proper ventilations in hot sunny days and quick draining of battery or decreasing the efficiency of vehicle. The paper is focused on increasing effectiveness of AC considering following parameters.

AC performance depends on geometry & operating parameter

1.2 Geometrical parameters and operating parameters

- Geometrical parameter
1. Duct shape and size
2. Cabin size
3. Windows area
4. Insulating properties of bus structure

- Operating parameter
1. Ambient temperature
2. Flow velocity of cooling spots in duct above passenger
3. Bus operating velocity and relative position of sun
4. Engine speed and internal heat generation

2. MESH SIZE DETERMINATION

2.1 Methodology to determine mesh size

Mesh size for CFD flow problem depends upon the turbulence model of CFD that has to be use, velocity of flow in model and use of wall function. For getting some initial data of all this we are running simulation with course mesh.

2.2 Initial CFD simulation

Mesh size for initial simulation is 50mm is solved initially with boundary conditions shown in fig.1 the initial Results shown below

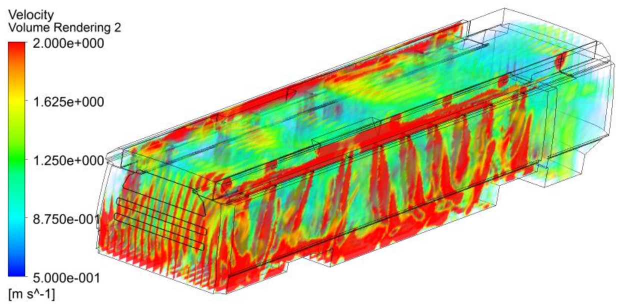

Fig -1: Velocity volume rendering of initial simulation

2.3 Data and calculations from data obtained by initial CFD simulation

Description	Max. Velocity of air	Importance in simulation
Passenger sitting area	15 m/sec	High
Inside duct	40 m/sec	moderate

Lobby area	5 m/sec	low
Driver cabin	8 m/sec	High
Around passenger windows	5 m/sec	High

Table -1: Velocity obtained by initial run of bus

2.3.1 Turbulence calculation

Velocity = 40m/sec

Characteristic length = 1 m

Kinematic viscosity of air = 1.4207E-5 m2/sec

Using

Reynolds number = ρVl/μ

Where

V= Velocity of the fluid

l= the characteristics length

μ/ρ= Kinematic viscosity

Reynolds number = 2815513

Since the Reynolds number is high thus flow can be considered as reliable for free-shear flows thus k-ϵ realizable turbulence model will be appropriate for the problem.

Generally when you building grid for RANS model of k-epsilon the target value of y+ should be between 30 and 300. The y+ depends on friction velocity u*=sqrt(Tw/rho) where Tw is wall shear stress, element distance to the nearest wall and local kinematic viscosity.

For this problem we are targeting y+ value of 100 as wall shear is not that important in our problem

2.3.2 Calculation for first cell height and 2D mesh size

Using

$C_f = f(R_e)$; $T_w = (C_f \rho U^2)/2$
$u^T = \mathrm{sqrt}(T_w/\rho)$; $\Delta y = y^+ \mu / u^T \rho$

Since allowed aspect ratio is 10 to 15 in first prism layer thus 2D mesh size will be equal to 10x of first cell thickness.

Description	V(m/sec)	First cell height	2D element size
Passenger sitting area	15	2.354 mm	23.54 mm
Inside duct	40	0.959 mm	9.59 mm
Lobby area	5	6.38 mm	63.8 mm
Driver cabin	8	4.17 mm	41.7 mm
Around passenger windows	5	6.38 mm	63.8 mm

Table -2: first cell height and 2D elements size calculated values

2. CFD SIMULATION CONSIDERATIONS

For simulation we used a bus that is commercially available whose geometry is defined below.

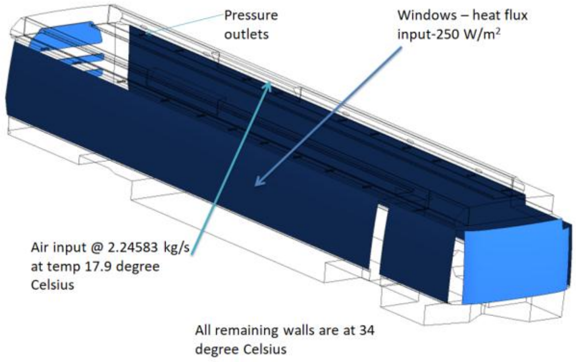

Fig -2: Boundary conditions for CFD of bus

Considering operating condition of bus in warm environment with direct application of sunlight and full capacity AC application the following parameters can be considered.

Description	Value	Reference
Ambient air temperature	34°C	Average temperature in warm day
Heat from windows	250 W/m²	Solar heat flux
Mass flow rate of cold air	2.24583 kg/s	Calculated below
Cold air temperature	17.9°C	Calculated below
Passenger manikin temperature	37 °C	Average human body temperature

Table -3: Boundary conditions

2.1 AC cold air flow in duct calculations

Inputs by ac manufacturer

Air flow of blower: 110 m³/min

Ac capacity: 9 ton

Heat rejection rate (watt) =10.34 ton × 3500=36389 J/sec

Mass flow rate of air = density of air × Volume flow rate

=1.225 kg/m³ × 110 m³/min

=134.78 kg/min = **2.24583 kg/s**

Heat rejection by air = mass flow ×specific heat capacity × temperature difference

36389 J/sec = 2.24583 kg/s × 1006.43 J/kg -˚C × temperature difference

Temperature difference = 16.1 ˚C

Cold air temperature = ambient temperature - Temperature difference

Cold air temperature = 34 ˚C - 16.1 ˚C = 17.9 ˚C

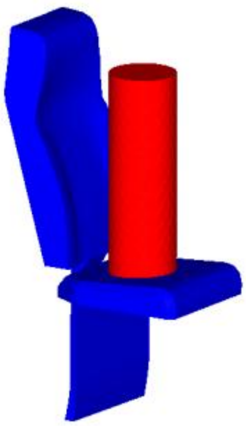

Fig -3: Geometry of manikin and seat in bus

2.2 Modelling for CFD

For modelling we uses hypermesh as preprocessor and created 2D shell mesh and for making polyhedral cells we used fluent.

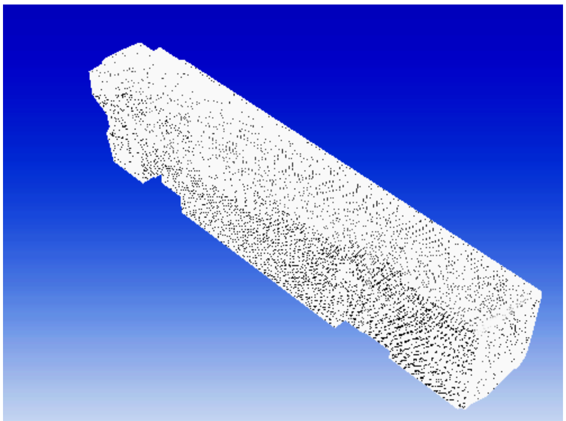

Fig -4: Polyhedral meshed model

Fig 5: Seating layout

2.3. **CFD SIMULATION RESULTS**

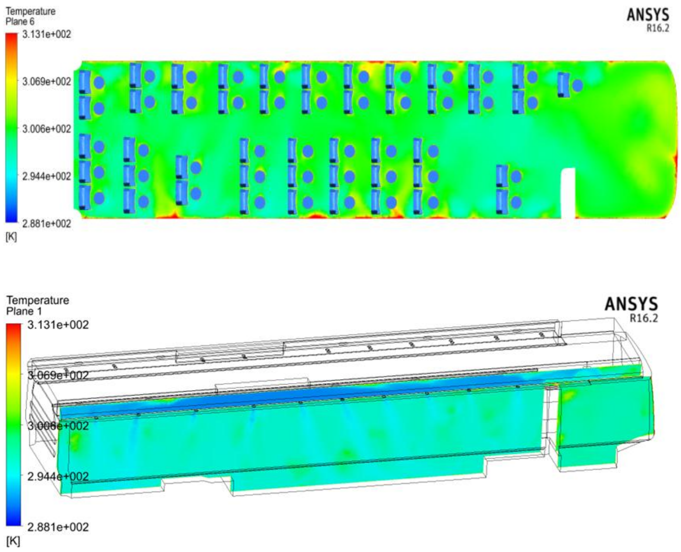

Fig 6. Contour of temperature

From the Contour one can easily identifies the region which is hotter like in middle of bus near windows we can identifies some discomfort in passenger. The same thing can be confirms if we plot horizontal sections of contour shown below.

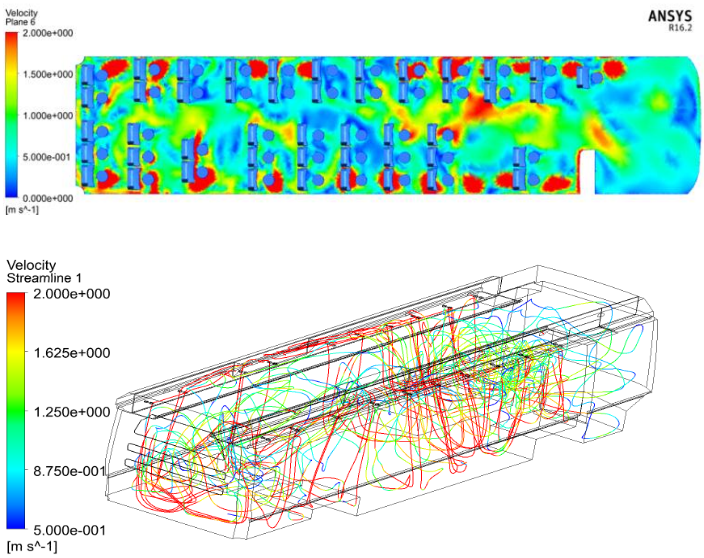

Fig 7. Contour of air velocity

From the above contour one can easily finds the middle portion has higher temperature and low air velocity thus passenger in the middle will feel less comfortable.

3. CONCLUSIONS

- The temperature around the occupant seems to be ok, somehow there are certain regions warmer near the windows and temperature around the middle region is not uniformly distributed.
- Air velocity in the middle of bus is quite low
- Modification in duct to provide higher flow of air in middle can solve the problem

ACKNOWLEDGEMENT

Dr. Pradeep kr Patil[5]

[5]*Professor,* Swami Vivekanand College of Engineering For giving huge support in research and development of model.

REFERENCES

[1] Parametric Investigation Using Computational Fluid Dynamics of the HVAC Air Distribution in a Railway Vehicle for Representative Weather and Operating Conditions PUBLISED IN Energies 2017, 10(8), 1074; https://doi.org/10.3390/en10081074

[2] Numerical Simulation of the Thermal Comfort in a Model of a Passenger Car Cabin, January 2016

[3] Notes on Numerical Fluid Mechanics and Multidisciplinary Design 132

[4] On the Possibility of CFD Modeling of the Indoor Environment in a Vehicle, Sustainable Solutions for Energy and Environment, EENVIRO 2016, 26-28 October 2016, Bucharest, Romania

[5] Nitin S. Gokhale, Sanjay S. Despande, Dr. Anand N. Thite, "Practical Finite Element Analysis", Finite To Infinite, India, 2007.

[6] 2-D CFD analysis of passenger compartment for thermal comfort and ventilation; International Journal of Engineering and Manufacturing Science. ISSN 2249-3115 Vol. 7, No. 1 (2017)

[7] Evaluation of Cabin Comfort in Air Conditioned Buses Using CFD ; 2014-01-0699

[8] Cfd Analysis Of Air Flow And Temperature Distribution In An Air Conditioned Car; International Refereed Journal of Engineering and Science (IRJES) ISSN (Online) 2319-183X, (Print) 2319-1821 Volume 2, Issue 4

[1] Numerical Simulation of the Flow in a Passenger Compartment and Evaluation of the Thermal Comfort of the Occupants; SAE Transactions; Vol. 106, Section 6: JOURNAL OF PASSENGER CARS: Part 1 (1997), pp. 806-816 (11 pages)

[2] Cfd analysis on aerodynamic effects on a passenger car; International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056 Volume: 04 Issue: 09 | Sep -2017

[3] Numerical Study on Human Thermal Comfort in a Passenger Train; Transactions of the Korean Society of Automotive Engineers

BIOGRAPHIES

Sagar Verma
ME Student,
Swami Vivekanand College of Engineering

Dr. Rahul joshi
Associate Professor,
Swami Vivekanand College of Engineering

Dr. Mayank Ladha
Associate Professor,
Swami Vivekanand College of Engineering

Mr. Manoj sharma
Assistant Professor,
Swami Vivekanand College of Engineering