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

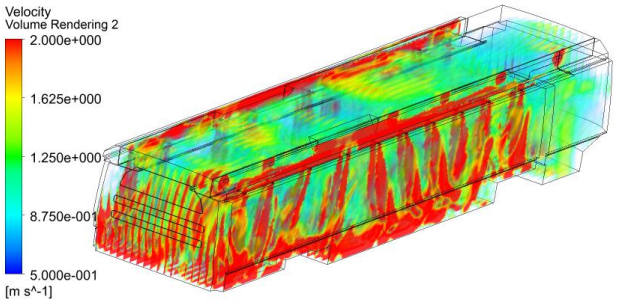

Fig -1: Velocity volume rendering of initial simulation

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