Numerical Simulation and Analysis of HVAC Duct

Student Name: Shivanand Doddaganiger **Guide Name: Dr. N. R. Deore**

Simulation of passenger compartment climatic conditions is becoming increasingly important as an alternate to wind-tunnel and field testing to achieve improved airflow comfort while reducing vehicle development time and cost. Computational Fluid Dynamics (CFD) analysis of a passenger compartment involves not only geometric complexity but also strong interactions of airflow. Temperature and velocities are major factors responsible for cabin temperature. Primary focus of the study is to analyse existing airflow and propose improvement in its duct shape and vent orientation for passenger comfort. Air-flow inside a vehicle cabin because of airflow distribution over manikin is also part of the study. Investigation of fluid flow through HVAC duct form different outlets of an automotive heating ventilation and air conditioning (HVAC) system will carry out in the present work. The CAD model was developed and analysis will be done. To analyse the air flow, a simulation is performed using Computational Fluid Dynamics, and with the help of this simulation we can roughly estimate the behaviour of air. The performance of the HVAC system is judged by parameters like air discharge rate at cabin level, pressure drop through the system, uniformity of the air flow at the outlet faces and distribution between different duct outlets. Pressure loss is another aspect which will require a lot of attention at this stage of development. It is one of the major characteristic which will ensure a smooth flow of air inside the HVAC system. All these parameters are predicted by computational fluid dynamics (CFD) analysis and they should meet the requirements. Comparison between CFD and testing results will be made in the HVAC system development by incorporating CFD as a design tool.

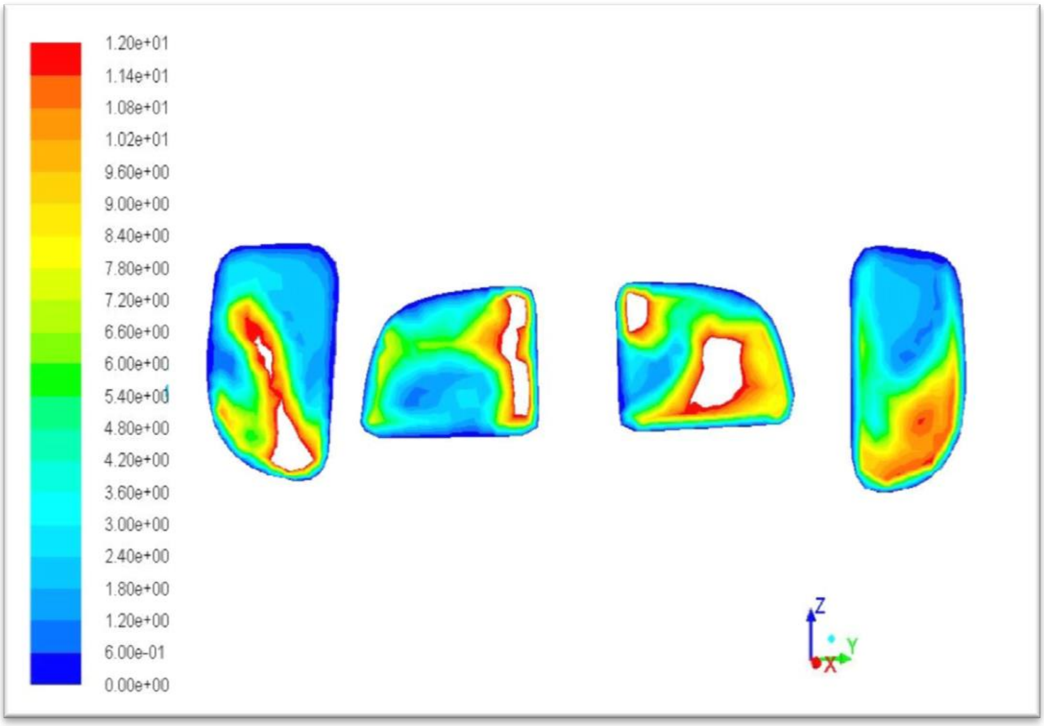

Fig. Velocity Contours at different outlets (m/s) with guides