Numerical Simulation and Flow Analysis of Rear Transfer Ducts Using 

**AcuSolve**

**Manu C S**  
*Engineer*  
*KPIT Technologies Ltd.*  
RMZ Ecoworld  
Bengaluru – 560103, India  
Manu.s@kpit.com

**Gunjan Keshri**  
*Technical Lead*  
*KPIT Technologies Ltd.*  
RMZ Ecoworld  
Bengaluru – 560103, India  
Gunjan.keshri@kpit.com

**Abbreviations:**  
CFD: Computational Fluid Dynamics  
HVAC: Heating, Ventilation and Air Conditioning

**Keywords:** HVAC duct, Air flow analysis, Numerical simulation, CFD.

**Abstract**

Flow analysis of passenger compartment in bus is becoming really important to understand the airflow inside HVAC duct. Current study is focused to assess the existing airflow in HVAC duct and flow distribution at multiple vents and to propose improvements in duct design, vent placements for passenger comfort.

In this effort, Altair HyperWork’s commercial CFD code, AcuSolve has been used to simulate rear transfer duct of bus. CAD model of HVAC duct were imported into HyperMesh for geometry clean-up and generating surface mesh. Volume mesh was generated in AcuConsole and solved using AcuSolve. Post-processing of the results is carried out using AcuProbe & AcuFieldView for better understanding and visualizing the flow fields inside ducts. Pressure drop through the system and uniformity of the airflow at outlet faces was optimized using CFD analysis.

**Introduction**

The application of HVAC for double deck buses have a common theme of high performance which plays an important role of passenger’s thermal comfort. To improve air conditioning performance and occupant thermal comfort, a thorough understanding of fluid flow in a HVAC system is required. With the development of huge computational power and numerical techniques, it is possible to conduct CFD simulations and to optimize for better performance without involving to invest more money on physical test procedures.

The present work involves the study of a numerical simulation of air mass flow distribution in rear transfer ducts. The geometry consists of four independent ducts as shown in Figure 1, two ducts are located in the upper saloon and other two are located in the lower saloon of a double deck bus. Individual duct varies in length and have multiple outlets. Based on the sectional area of the vents, they have been categorized as large and small outlets to define the boundary condition.

Rated mass flow of blower was defined at the inlet of each duct and the flow distribution at multiple outlets were monitored. Low system pressure loss, uniform air flow over outlet faces, proper air flow split at the outlet vents are some of the criteria considered in the optimization of HVAC system.
Process Methodology

Pre-processing involves necessary geometry cleanup, extracting fluid contact surface of rear transfer ducts as shown in Figure 2. Surface mesh creation using HyperMesh, followed by tetrahedral mesh generation along with boundary layers using AcuConsole.

The boundary conditions were defined on the surface patch as shown in Table I. Problem was setup with air as the fluid material, steady state analysis and Navier-Stokes equations were solved to determine the flow fields. Spallart-Allmaras turbulence model was accounted for capturing turbulence effects.

**Table I: Boundary Conditions**

<table>
<thead>
<tr>
<th>Surface</th>
<th>Boundary Name</th>
<th>Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>Inflow</td>
<td>Mass Flux</td>
</tr>
<tr>
<td>Outlet</td>
<td>Outflow</td>
<td>Static Pressure - 0 gauge pressure</td>
</tr>
<tr>
<td>Duct, Airway</td>
<td>Wall</td>
<td>Wall - no slip</td>
</tr>
</tbody>
</table>
The convergence status during steady state calculation was monitored using AcuProbe for the minimum residual values to reach one thousandth. Mass balance was also monitored along with the flow convergence to check equilibrium.

With the help of AcuProbe, percentage of mass flow distribution at multiple outlets was plotted. Figure 3 and Figure 4, shows the percentage of mass flow distribution of Rear Transfer Duct at larger outlets and smaller outlets respectively.

**Figure 3**: Percentage of Mass Flow Distribution at Larger Outlets

**Figure 4**: Percentage of Mass Flow Distribution at Smaller Outlets
AcuFiledView was used to post-process the results. Pressure contour and velocity streamline of Rear Transfer Duct is shown in Figure 5 and Figure 6 respectively.

*Figure 5: Pressure Contours of Rear Transfer Ducts*

*Figure 6: Velocity Streamlines of Rear Transfer Ducts*

In the Figure 3, Lower Saloon Right Duct model shows negative flow at initial outlet vents. This is because of its position and abrupt geometry, has tendency to create local low pressure zone which in turn tries to suck air inwards. This similarity was also found in Upper Saloon Left Duct and is observed in Figure 4. These locations of flow recirculation were identified, and are shown in the Figure 7 and Figure 8.
Figure 7: Flow Recirculation in larger outlets of Lower Saloon Right Duct

Figure 8: Flow Recirculation in smaller outlets of Upper Saloon Left Duct

Results & Discussions
The numerical analysis of Rear Transfer Ducts for air flow distribution in Rear Transfer Ducts was analyzed using AcuSolve. Total system pressure drop for the individual duct was calculated and is reported in the Table II. Lower Saloon Right Duct has highest system pressure drop among all other ducts. This is because of its geometry configuration, causes obstruction to the flow which results in higher pressure drop.

<table>
<thead>
<tr>
<th>Sl. No.</th>
<th>Rear Transfer Ducts</th>
<th>System Pressure Drop</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Upper Saloon Right Duct</td>
<td>17.5 %</td>
</tr>
<tr>
<td>2</td>
<td>Upper Saloon Left Duct</td>
<td>46.2 %</td>
</tr>
<tr>
<td>3</td>
<td>Lower Saloon Right Duct</td>
<td>243.5 %</td>
</tr>
<tr>
<td>4</td>
<td>Lower Saloon Left Duct</td>
<td>92.6 %</td>
</tr>
</tbody>
</table>
Benefits Summary

1. CFD analysis using AcuSolve was helpful to foresee the performance of Rear Transfer Duct before adapting or execution in systems.
2. AcuSolve CFD code is architected for parallel computing and memory distribution which provides fast and efficient steady state solutions for standard unstructured element topologies.
3. CFD modelling and solving in AcuSolve procures reliable result. In the present study, it saves cost by 30% and turnaround time by 70%.

Future Plans

In the present study, we have simulated for air flow distribution in HVAC ducts considering Spalart-Allmaras model which solves a single conservation equation for turbulent viscosity. To make our results more realistic, we will simulate considering two equation turbulence model to predict better pressure gradients, strong streamline curvature, swirl and rotation.

Conclusions

Baseline model was analyzed to understand the air flow distribution inside Rear Transfer Ducts. Based on the results obtained, Lower Saloon Right Duct has higher system pressure drop compare to other ducts. Key locations which are responsible for pressure drop have been identified and suggested for design alternation in order to improve the overall flow.

ACKNOWLEDGEMENTS

I would like to express my thanks to our project account manager Mr. Radhakrishna S, for his support to make this thing possible. I would like to thank CAD team who are involved directly or indirectly in this work. Also I owe my deep gratitude to Altair team for their technical support and guidance.

REFERENCES

[2]. Mr. Shivanand and Mr. Narendra, “Numerical Simulation and Analysis of HVAC Duct.”
[3]. Dr. Neihad Al-Khalidy, “Design optimization of Industrial Ducts Using CFD.”
[4]. H K Versteeg and W Malalasekera, “An Introduction to Computational Fluid Dynamics”.
[5]. Altair Hyperworks resource library and technical papers.