Optimizing an Automotive Air Handling Unit for Uniform Temperatures using FloEFD™

By Lu Ping, Pan Asia Technical
Automotive Center, Shanghai, China

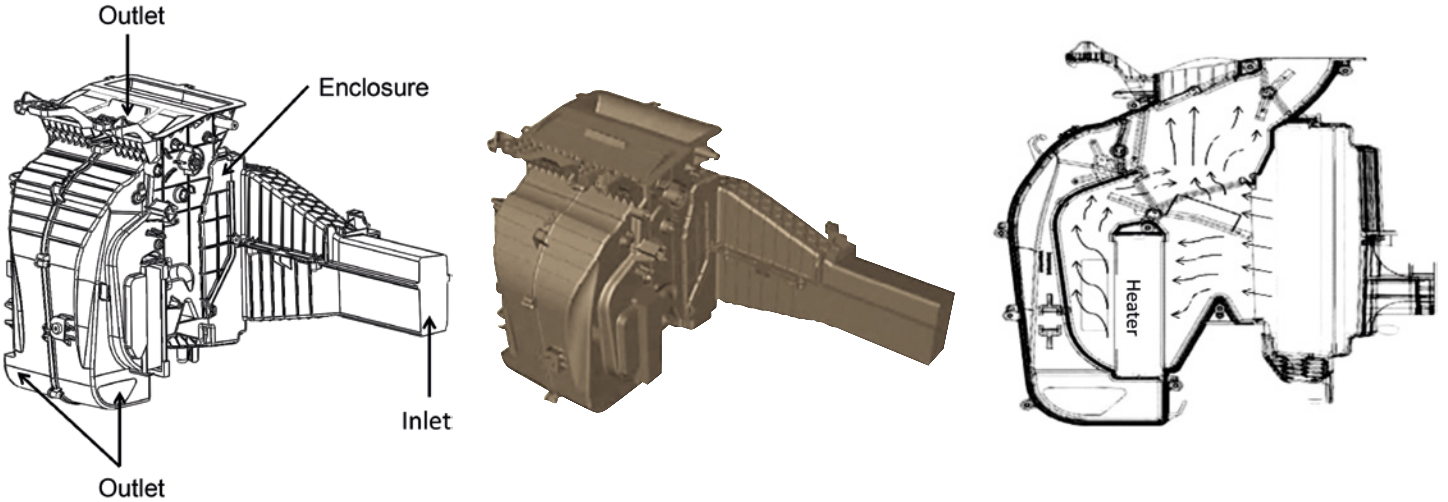
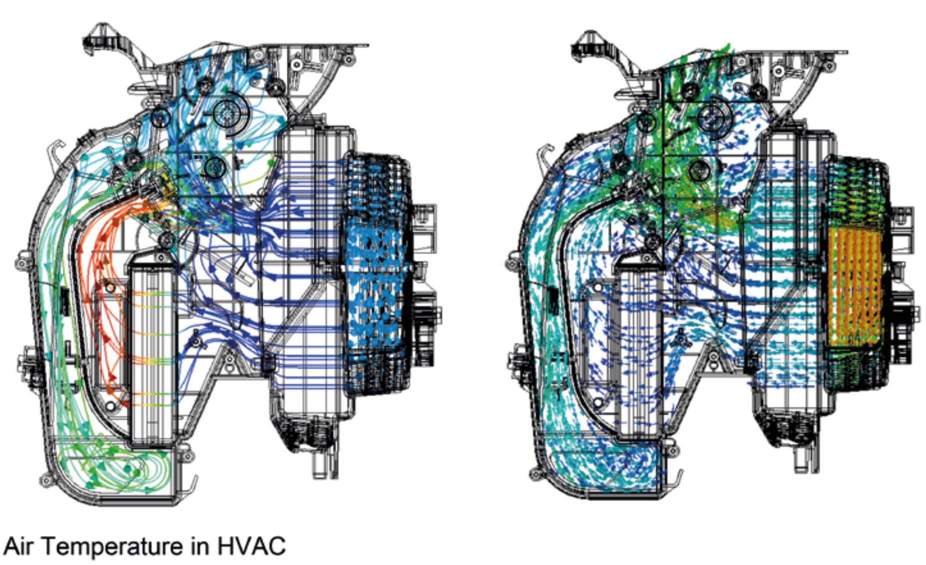


Figure 1. AHU geometry, its CAD Model, and a Sectional Schematic of Airflow Paths through it

The number of car owners in China is increasing exponentially. China will soon have nearly as many drivers as the U.S. With this band of newly qualified drivers, a demand for higher standards in vehicle ride "comfort" is developing. One such area is the standard of cabin comfort. This is directly related to a car's air-conditioning unit with discharge temperature uniformity which is one of the key factors impacting perceived comfort levels.

On the one hand, discharge air temperature from the HVAC air box has to be uniform for passenger comfort, but on the other, uniformity can reduce the extent of the automatic air-conditioning calibration workload. However, due to packaging limitations in typical vehicle development, its air conditioning unit has to be as compact as possible, which usually make it a poor or inadequate mixture of cold and hot airflow inside the air conditioning unit and finally leads to an non-uniform discharge temperature. In the development of automotive HVAC air handling units (AHU), to control the discharge air temperature uniformity, performance is



Air Temperature in HVAC
Lowering the air temperature by Evaporator
Heating the air temperature by Heater
The air temperature is dependent on Throttle in different Position
The function of parametric study in FloEFD is useful for to solve several scenarios.

Figure 2. FloEFD predictions of airflow Vectors (right) and Temperature Distribution (left) inside the AHU

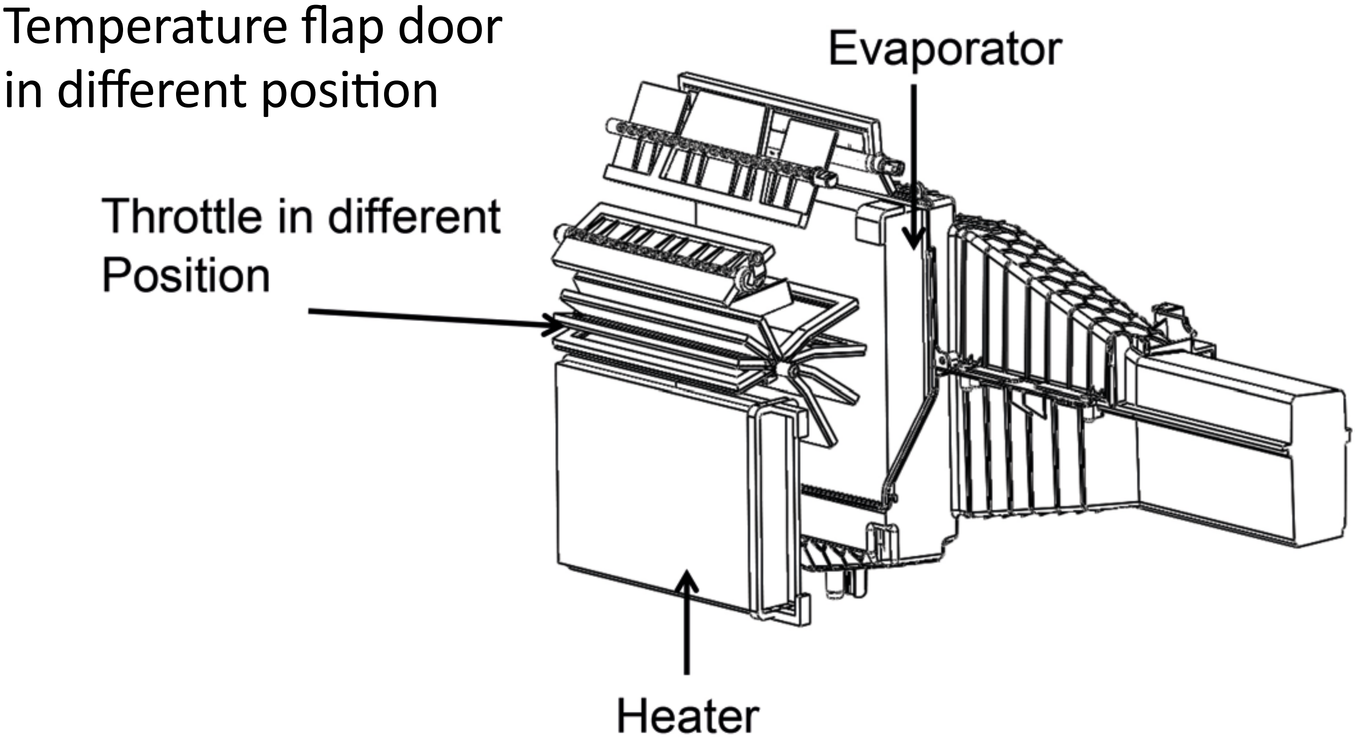


Figure 3. Air Handling Unit geometry showing details of the Evaporator, Heater and Temperature flap door under different positions

key, and it is important to consider the factors mentioned above for the development of a car's HVAC air handling unit (AHU).

Figure 1 shows the specific AHU being evaluated in this study. Flow through it involves complex tortuous passageways and the mixing of both cold and hot airflows. The unit has one inlet and two outlet zones, and its complex geometrical nature means that it is most realistic to simulate fluid flow and heat transfer inside a CAD package using a CFD tool such as FloEFD. The AHU itself consists of air box housing, an evaporator, a heater, and flap door components. During normal operation, airflow enters the air conditioning unit through the intake housing, and then flows through the evaporator to be cooled down. After cooling, the airflow partially goes through the heater core to be warmed up while part goes towards the outlet area with the flow guiding of a temperature flap door. These two hot and cold air streams then re-converge and mix to achieve a proper and comfortable temperature. Conditioned airflow is finally delivered to passengers through the air box outlet. A typical FloEFD simulation prediction for airflow vectors and temperature effects inside the AHU is shown in Figure 2.

The position of the temperature flap door effectively acts as a control valve inside the unit and ultimately determines the hot and cold airflow "mixing ratio". It can be altered to different positions (Figure 3). The "hedge angle" and "area ratio" of the cold and hot airflow channel have an important influence on the final mixed airflow temperature distribution.

	Runner Hedge Angle (°)	Runner Area Ratio (%)
Case 1	120	44
Case 2	120	49
Case 3	120	39
Case 4	116	44
Case 5	116	49
Case 6	116	39
Case 7	124	44
Case 8	124	49
Case 9	124	39

Table 1. The nine AHU CFD Simulation Scenarios examined in this study

The CFD boundary conditions simulated in this AHU study extended from airflow rates of 15l/s to 60l/s at an air inlet temperature of 20°c with 875W heat transfer rate from the heater component. Nine parametric CFD simulations inside FloEFD were used to determine an optimized cold and hot flow channel "hedge angle" together with runner "area ratios" as shown in Table 1.

This parametric study focused on the AHU outlet airflow temperature distribution under different temperature flap door setting. More focus is around the middle position, that is, for angle degree of outlet damper door from 25° to 50°, considering the middle position is relative to a customer's actual high frequency usage scenario (see Figure 4). The temperature difference is seen to be optimal for Case 3 for the two temperature flap door conditions. Hence, the cold and hot airflow channel hedge angle and runner ratio area under this case is the most ideal which was verified visually (Figure 5) by outlet CFD temperature contours under these two temperature flap door positions.

Finally, we validated the CFD simulation Case 3 prediction against an experimental test of the actual car AHU. We chose an air conditioning box inlet temperature of 0°C, and the heater inside operating with a 90°C fluid so as to replicate a real vehicle use of air conditioning over cold and hot atmospheric conditions. By adjusting the temperature flap door in the AHU to control air-conditioning of cold and hot air mixing, we were able to verify the box's linear temperature uniformity performance target. We positioned 4 thermocouples on each outlet and measured the average exit air temperature. Figure 6 shows the actual measured performance data of the AHU. Aligned with the CFD simulation results, we achieved the maximum temperature difference within 4°C among four vent outlets when the temperature flap valve is adjusted between 35% and 65%. We reached the requirement of a linear thermal design, while at the same time it was basically consistent with the virtual design CFD results.

In conclusion, we adopted the commercial CFD software, FloEFD, for this study because of its ease of use in meshing when compared to the tetrahedral or prismatic meshing approaches in traditional CFD codes. We found that FloEFD gives more accurate and more efficient CFD simulation results. Since it works within the mechanical CAD environment, it is a highly engineered universal fluid flow and heat transfer analysis software. FloEFD was able to examine a range of AHU hedge angles for hot and cold airflow channels. The hedge angle and area ratios of 120° and 39% respectively were found to be the most optimal. FloEFD with its parameterized calculation function was highly efficient in varying a range of AHU parameters that we studied. It showed great design performance improvements in terms of achieving an optimized design while at the same time reducing our overall cost of development.

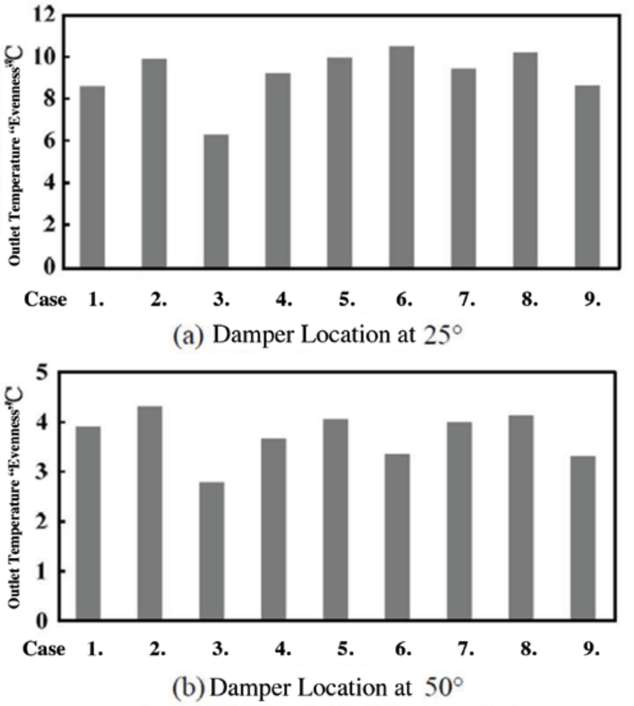


Figure 4. CFD predictions of Outlet Air Temperature "Evenness" for the nine different hedge ratio Cases at two Temperature Flap Door Angles

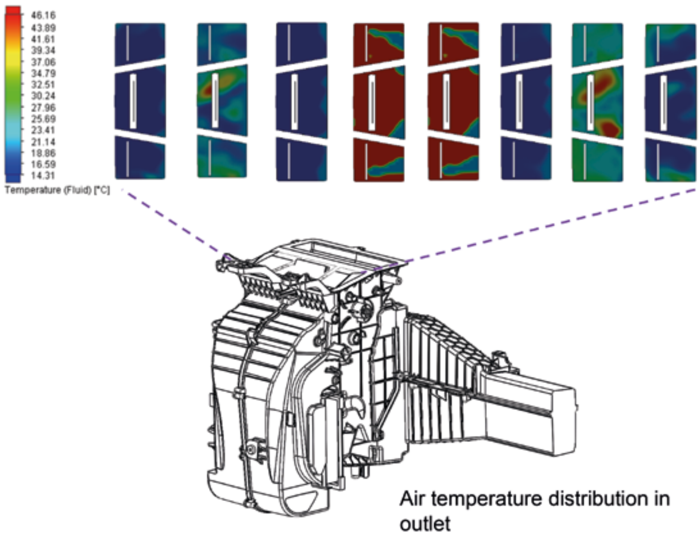


Figure 5. Air Handling Unit predicted Air Temperature Contours in the outlet face for the different hedge ratios

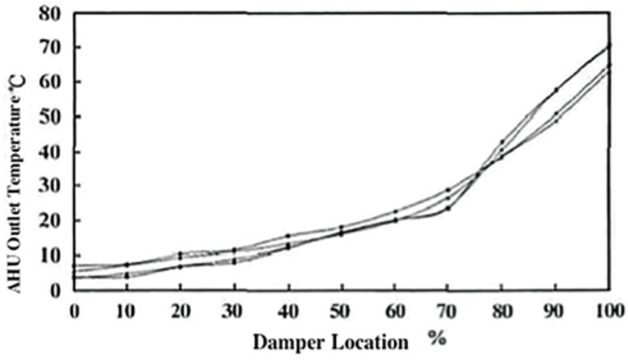


Figure 6. Data from four experimental Thermocouples of Outlet Air Temperature versus Temperature Flap valve Location for Case 3