Simulation Analysis of Flow Field in Engine Compartment of a Rear-Mounted Exhaust

To cite this article: Y Y Xie *et al* 2019 *IOP Conf. Ser.: Mater. Sci. Eng.* **562** 012056

Simulation Analysis of Flow Field in Engine Compartment of a Rear-Mounted Exhaust

Y Y Xie[1], L Q Zhang[1,*], Y L Yan [1], S W Qiu [2] and C Z Tan [3]

[1]Mechanical and Electrical Engineering Institute, Central South University of Forestry and Technology, Changsha, 410000, Hunan, China
[2]State Key Laboratory of Advanced Design and Manufacture for Vehicle Body, Hunan University, Changsha, 410082, Hunan, China
[3]Changan Oushang Automotive Research Institute, Chongqing, 400023, China
[*]Corresponding author e-mail: 21384881@qq.com

Abstract. The numerical simulation of the flow field in the engine compartment of a car is the basis for the evaluation and research of the air flow characteristics in the engine compartment. In this paper, the computational fluid simulation software STAR-CCM+ was applied to simplify the vehicle model of a rear-mounted exhaust. The surface wrapper and surface remesher were used to obtain the whole vehicle surface mesh. Combining with the vehicle wind tunnel model was generated the volume mesh based on Trim mesh sub-regional method. Then, the finite element model of the car engine compartment was established. Three-dimensional flow field in engine compartment was analyzed under reasonable boundary conditions, and found the reflow at climbing is more apparent as compared with that at high constant speed. Then the flow rate experiment of the whole vehicle was obtained wind speed value of the windward surface of the radiator. The error between the simulated value and the experimental value is within 10%, indicating that the simulation model established in this paper is reliable and possesses high simulating accuracy.

1. Introduction

Engine compartment structure is very complicated, and the flow characteristic of engine compartment is closely related to its service life and performance. Therefore, it is extremely important to explore an accurate finite element modeling method for the thermal management of engine compartment. In recent years, lots of relevant researches have been conducted at home and abroad [1-3]. Previously, the finite element model of engine compartment is built by the simplification of vehicle and followed by tetrahedral mesh. Then the flow field in the engine compartment is calculated [4-6].

In present work, the vehicle wind tunnel model with detail feature of the main components is built based on subregional Trim mesh, and the vehicle flow fields at climbing and high constant speed conditions are simulated by the corresponding boundary condition setting. Then, the flow field distributions of simulation in the engine compartment are analyzed, and the accuracy is verified by the vehicle flow velocity experiment.

2. Engine Compartment Model

2.1. Geometric Model Simplification

This paper takes a rear-mounted exhaust model as the research object. The ratio of the three-dimensional geometric model to the actual car size is 1:1. The length, width and height of the vehicle are 5300 mm, 1800 mm, and 1470 mm respectively. The engine compartment mainly includes

IOP Conf. Series: Materials Science and Engineering **562** (2019) 012056 doi:10.1088/1757-899X/562/1/012056

components such as cooling modules (radiator, condensers and fans), engine blocks, transmissions, oil piping, electrical piping and cables.

In order to improve the calculation efficiency and reduce the amount of calculation. In the software STAR-CCM+, parts with little effect on the flow field are removed, holes with diameter less than 10 are filled and connecting positions of door, window and other sheet metal are sealed. The processed engine compartment model is shown in figure 1.

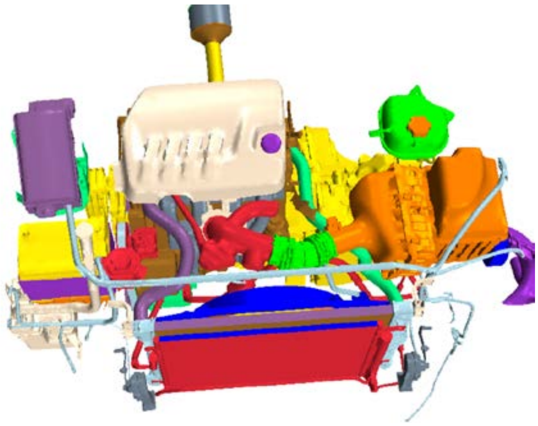

Figure 1. The processed engine compartment model.

2.2. Meshing

2.2.1. Surface meshing. Based on the processed model, the surface wrapper is carried out by different sizes setting according to the part characteristics. Then, the topologically closed model is obtained for each component. The surface remeshing is conducted to meet the requirement of face quality and face proximity. The remeshed engine compartment and whole vehicle models are shown in figure 2 and figure 3.

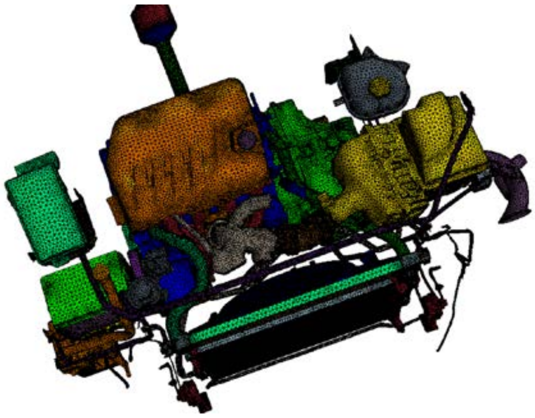

Figure 2. The remeshed engine compartment model.

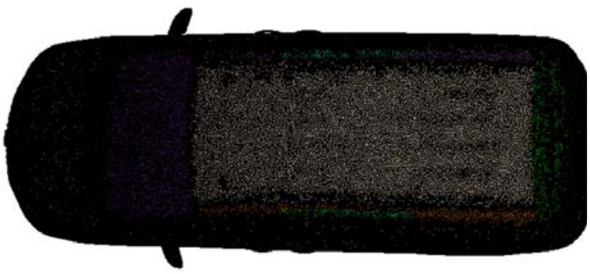

Figure 3. The whole vehicle model.

In theory, the actual domain of external flow field is an infinite region. However, the wind tunnel calculation domain in practical engineering application usually adopts a regular rectangular parallelepiped. The front and rear spaces of the vehicle in the calculation domain take three and eight times the length of the vehicle, the upper and side spaces take five times of the vehicle height and three times of the vehicle width, respectively. The wind tunnel model of the whole vehicle is shown in figure 4.

Figure 4. The vehicle wind tunnel model.

2.2.2. Volume meshing. The volume meshed model of the wind tunnel calculation domain is obtained by Trim mesh based on the above wind tunnel surface model with a maximum size of 512 mm and a minimum size of 4 mm. The boundary layer adopts a prism layer mesh, and the wall surface expands outward to generate one layer mesh with thickness of 1 mm. In order to improve the simulation accuracy of the engine compartment flow field, the entire calculation domain is divided into 6 regions (engine compartment, chassis, vehicle body, calculation domain 1~3) and then meshed separately. The sizes in engine compartment, chassis , vehicle body, calculation domain 1~3 are 16 mm, 16 mm, 32mm, 64mm, 128mm, 256 mm, respectively. The mesh distribution of the calculation domain in the symmetry plane and a magnified section outlined in the circle are shown in figure 5.

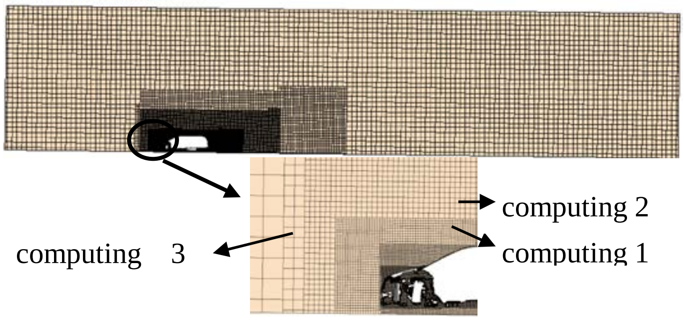

Figure 5. The mesh distribution of the calculation domain in the symmetry plane and a magnified section outlined in the circle.

2.3. Boundary Condition

2.3.1. External flow field boundary. Considering that the vehicle speed is much lower than the sound speed and the surrounding air density is constant, the calculation domain can be regarded as a three-dimensional incompressible flow field. The engine compartment flow field can be calculated based on the realizable k-ε turbulence model. The boundary of the wind tunnel entrance is set to Velocity-inlet, the set condition is the vehicle speed, and the exit boundary is set to the Pressure-outlet. This paper selects two conditions of climbing and high speed. The velocity of climbing and high speed is 80km/h, 160km/h, respectively.

2.3.2. Internal flow boundary. The internal structures of the condenser and radiator are complex, which are generally regarded as a constraint boundary and replaced by porous region model during simulation. The porous inertial resistance and the porous viscous resistance of the condenser and radiator are obtained by fitting the measured wind speed-pressure drop and Darcy law. The parameters of the porous region model are shown in table 1.

Table 1. Parameters of porous region.

Component	Porous inertial resistance(kg/m4)	Porous viscous resistance(kg/m3-s)
Radiator	131.926	853.333
Condenser	164.839	567.742

IOP Conf. Series: Materials Science and Engineering **562** (2019) 012056 doi:10.1088/1757-899X/562/1/012056

In present work, the MRF multiple reference system model is applied to simulate the rotation of fan. Accordingly, the fluid region around the fan blade is set to the rotating coordinate system. The rotation velocities of fan at climbing and high constant speed are set to 2000 rpm and 3500 rpm, respectively.

3. Results

3.1. Flow Field in the Engine Compartment

Velocity vector distributions of the engine compartment in the symmetry plane at climbing and high constant speed are shown in figure 6 and 7. As can be seen, the reflow at climbing is more apparent compared with that at high speed, showing a relatively larger reflow area.

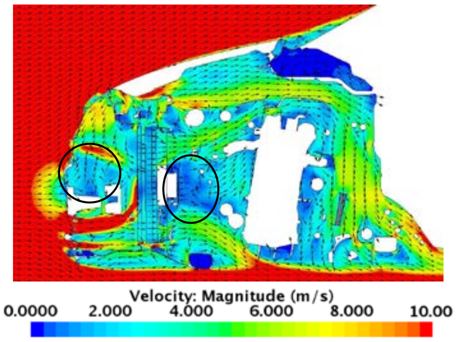

Figure 6. Velocity vector distributions of the engine compartment in the symmetry plane at climbing.

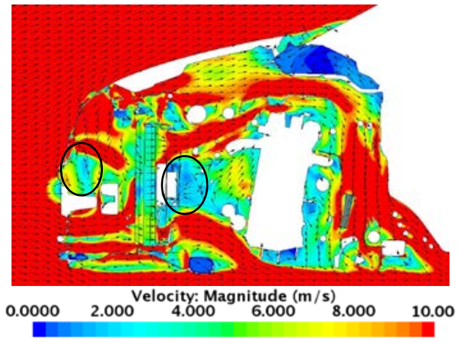

Figure 7. Velocity vector distributions of the engine compartment in the symmetry plane at high constant speed.

3.2. *Radiator Windward Surface Velocity*

Velocity distributions of the windward surface of the radiator at climbing and high constant speed are shown in figure 8 and 9. The measuring points are marked as 1, 2, 3 and 4. At climbing condition, the wind speeds of the measuring point are 1.22m/s, 2.65m/s, 2.11m/s and 2.58m/s, respectively. The average air velocity on the windward is 1.446m/s and the corresponding air volume is 0.502 kg/s. At high constant speed, the wind speeds of the measuring points are 4.46m/s, 5.23m/s, 4.22m/s and 5.64m/s, respectively. The average air velocity and the corresponding volume are 3.389 m/s and 1.167 kg/s, respectively.

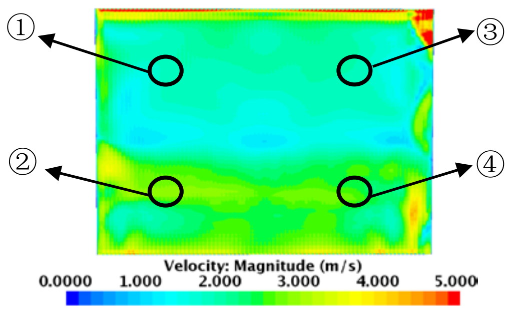

Figure 8. Windward flow velocity map of the radiator under the climbing condition.

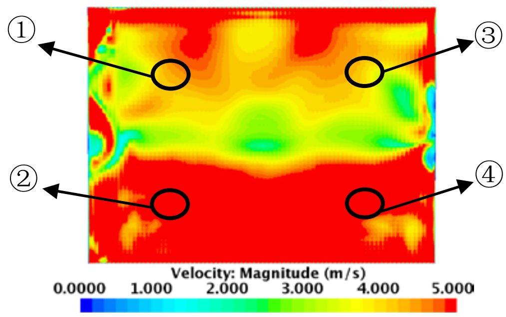

Figure 9. Windward flow velocity map of the radiator under high speed conditions.

4. Experiment Verification

In order to verify the reliability of the simulation results, the flow velocity of the points in Figs. 8 and 9 at climbing and high constant speed condition are measured by flow velocity test on the whole vehicle. The measuring points of the experiment are shown in figure 10 The locations are consistent with the simulation.

Figure 10. Flow velocity test on whole vehicle.

The measured flow velocity of the four points, and the percentage of the deviation between simulation and experiment at climbing and high constant speed conditions are shown in table 2. As can be seen, the maximum error is less than 10%, which indicates that the simulation model established in this paper is reliable and possesses high simulating accuracy.

Table 2. Simulation and experimental flow velocities of the specified points.

Conditions	Measured points	Simulation values (m/s)	Experimental values (m/s)	Error (%)
Climbing	1	2.21	2.46	-10.2
	2	2.65	2.90	-8.6
	3	2.11	1.95	8.2
	4	2.58	2.38	8.4
High speed	1	4.46	4.15	7.5
	2	5.23	4.95	5.7
	3	4.22	3.87	8.8
	4	5.64	5.96	-5.4

5. Conclusion

(1) The reflow at climbing is more apparent compared with that at high constant speed. This shows the reflow area is larger.

(2) The maximum error between the simulated value and the experimental value at climbing and high constant speed conditions is less than 10%. This indicates that the simulation model established in this paper is reliable and accurate.

6. Acknowledgements

The authors gratefully acknowledge research support from the Natural Science Foundation of Hunan Province (No. 2018JJ2681).

7. References

[1] Wang D and Han W 2015 Research and optimization of automobile engine compartment heat dissipation performance *Automobile Technology*. **12** 34-40

[2] Liu C B, Yang Y, Mo Y M and Wang W 2018 Simulation and optimization on vehicle underhood heat dissipation performance *Mechanical Design & Manufactu*re. **12** 137-140

[3] Jan J Y, Srinivasan K, Sun R L and Gleason M E 2000 Rapid simulation methodology for under-hood aero/thermal management *International Journal of Vehicle Design*. **1/2** 109-123

[4] Hai C. 2014 *Simulation research of heat dispersion of vehicle underhood* (Liaoning: Liaoning University of Technology)

[5] Sheng J W. 2017 *Truck Thermal management research and improvement by applying joint 1D/3D and coupled thermal simulation* (Chongqing: Chongqing University)

[6] Ni J M, Shen K, Zhu L M, Xu J H and Xu X Y 2013 Performance analysis of the cooling package used in rear-mounted engine bus in complete vehicle environment *Automobile Technology* **10** 35-39