Transient simulation of heat transfers for vehicle exhaust system

Guoquan Xiao*

School of Mechanical & Automotive Engineering, South China University of Technology, Guangzhou, 510640, China

Abstract

Transient simulation of heat transfers for vehicle exhaust system plays an important role in vehicle development process. A vehicle thermal management system model, which includes the underhood, the exhaust system and underbody, was established. In this model, the complicated boundary conditions between fluid and solid, which need to be specified in the non-coupled models, are now part of the solution and do not need to be specified. A transient simulation was undertaken for the heat transfer characteristics of this vehicle exhaust system. The heat transfer characteristics with the periodic pulsating movement of the exhaust gas velocity and temperature were obtained and analyzed, and the transient mean value were compared with the steady results, and the discipline between the steady heat transfer characteristics and the periodic pulsating movement of the exhaust gas velocity and temperature were found.

Keywords: Exhaust system; Thermal management; Fluid-solid coupling; Transient simulation

1. Introduction

Transient simulation of heat transfers for vehicle exhaust system has become an important part of the vehicle development process. About 1/3 of the fuel energy is carried away from the engine in the form of enthalpy and chemical energy with the exhaust flow [1]. The high temperature will impact the exhaust system component itself and its surrounding environment. When components are positioned too close to high temperature heat sources, such
as the exhaust manifold, overheating can occur. In addition, cooled air from the radiator can quickly pick up heat once again as it passes over heat generating components nearby.

| Nomenclature          | Description                                      |
|-----------------------|--------------------------------------------------|
| A                     | radius of                                        |
| B                     | position of                                       |
| C                     | further nomenclature continues down the page inside the text box |
| \( \dot{u}_{in} \)    | transient velocity of exhaust gas                |
| \( u_s \)             | mean velocity of exhaust gas                      |
| \( u_{max} \)         | maximal pulsating velocity of exhaust gas         |
| \( \omega \)          | pulsating frequency of exhaust gas                |
| \( T_{trans} \)       | transient temperature of exhaust gas              |
| \( T_{ave} \)         | mean temperature of exhaust gas                   |
| \( T_{max} \)         | maximal pulsating temperature of exhaust gas      |
| \( \phi \)            | flow parameter                                    |
| \( \Gamma \phi \)     | diffusion coefficient                             |

It is necessary to use computational dynamic fluid (CFD) codes to estimate vehicle exhaust system transient temperatures. There are essentially two methods, in the first method, measured heat flux data on exhaust pipes surface are used as the boundary condition to CFD simulations. While the prediction of exhaust pipes temperatures remains a largely test-driven process. Several examples of previous studies may be found in the literature, using CFD packages to compute heat transfer in vehicles [2-4]. The most detailed is due to Bendell, E. where a coupled steady-state CFD and thermal study was undertaken at full-vehicle scale [5-6]. Complete vehicle analyses have been made in a single vehicle model using Fluent code by Skea et al but correlations of surface temperature with test data were not published [7]. Obviously, it will not be possible to take this approach for new engine development whose parameters are beyond the range of the existing database. The second method is to calculate the temperature by the method of conjugate heat transfer. In many CFD simulations of this type, most models don't include radiation, which is the main mode of heat transfer between the exhaust system and components close to it [8-10]. Without radiation in the model, the temperatures need to be entered into the model, and this requires experimental data and the additional assumption of uniform values. Therefore, the ability to calculate transient heat transfer of vehicle exhaust system as accurately as possible is necessary. A transient analysis method for vehicle thermal management is presented and the temperature curve of a thermally-loaded part changing with different working conditions is calculated by coupling RadTherm with Fluent, the feasibility of transient thermal analysis is verified by test [11].

The paper is organized as follows: the next section describes the transient computational model. The conjugate heat transfers for vehicle exhaust system are described in the results section. Conclusions and discussions are given in the final section of the paper.

2. Transient Computation Model

One limitation of many vehicle exhaust system CFD analyses to date is that the size of the computational domain is unwieldy. CFD analyses are therefore often broken down into three separate regional simulations: front-end, underhood, and underbody. The boundary conditions for these submodels are either taken from experimental data or experience. A full sedan model surface data was supplied by CAD data and partly by a hypermesh model, in the computational model includes all heatshields, ground, side-walls, tunnel, dash, underfloor cover panels, fuel tank, exhaust pipes, resonators, muffler, tire-tub, rear-suspension and some other components that have dimension of 80mm or larger. Altogether, more than 150 components were included in a typical model, and the model structure is quite flexible and it allows easy addition of other vehicle components if they are deemed thermally important.
2.1 Computational domains

The resulting model is assumed to be positioned inside a wind tunnel of 35m in length, 16m in width, and 10m in height. The car model and computational domains used is shown in Fig.1.

![Fig.1. The car model and computational domain.](image)

2.2 Computational mesh model

The vehicle surfaces were discretized with triangle meshes. Surface mesh quality is modified by controlling skewness 0.85. Tetrahedral cells were used to discretize the fluid and the solid in the computational domain, and the volume mesh quality is modified by controlling mesh skewness 0.95, there is shown in Table 1.

| type       | Fan | Heat exchangers | Exhaust gas | Pipe wall | External airflow | Total |
|------------|-----|----------------|-------------|-----------|-----------------|-------|
| Grid       | 300 | 70             | 100         | 80        | 2,850           | 3,300 |

2.3 Boundary Conditions

The periodic pulsating movement of the exhaust gas were given as follows Eq (1) and Eq (2):

\[
u_{in} = u_s + U_{\max} \sin \omega t
\]

\[
T_{in} = T_s + T_{\max} \sin \omega t
\]

The periodic pulsating values of the exhaust gas velocity and temperature were coded into subroutine UFILE, which can be called by STAR-CD software [12], and the time step was given by the engine speed. The other boundary conditions can be represented by the steady simulation at vehicle uphill condition with full load [1].

2.4 Numerical methods

The conjugate heat transfer simulations were carried out by using STAR-CD which solves the equations as follows Eq (3):

\[
\frac{\partial}{\partial x} (\rho u \varphi) + \frac{\partial}{\partial r} (r \rho u \varphi) + \frac{\partial}{\partial \theta} (\rho u \varphi) = \frac{\partial}{\partial x} \left[ \rho \frac{\partial \varphi}{\partial x} \right] + \frac{\partial}{\partial r} \left[ \rho \frac{\partial \varphi}{\partial r} \right] + \frac{\partial}{\partial \theta} \left[ \rho \frac{\partial \varphi}{\partial \theta} \right] + s_v
\]

To model the heat transfer to the air from the cooling pack, the condenser and radiator flow resistances were modelled as porous media. The heat transfer was estimated approximately based on the data from the respective suppliers of AC condenser, radiator and fan. The enthalpy sources were coded into subroutine SORENT and were added to the fluid energy equation, representing the effects of heat exchangers on the fluid thermal field.

The velocity and temperature of the condenser and radiator were monitored and the simulations were stopped when the periodic pulsating values of these results were found.
3. Results of Transient Heat Transfer

The transient fluid-solid conjugate heat transfer characteristics with the periodic pulsating movement of the exhaust gas velocity and temperature were obtained, the qualitative trend of flow field and the thermal field predicted by these analyses is reasonable. The transient results of the current analyses are presented in figure forms.

The transient airflows with the periodic pulsating exhaust gas velocity and temperature are presented in Fig.2 (Y=80 mm) and Fig.3 (Z=150 mm) and analyzed quality in a circle (From 0.5S to 0.6S).

![Fig.2. Temperature of y=80 mm plane in a circle.](image1)

![Fig.3. Temperature of z=150 mm plane in a circle.](image2)

As can be seen in Fig.2 and Fig.3, the predicted external airflows pulsating temperatures are lower than 10°C, and the predicted exhaust gas pulsating temperatures are higher than 50°C, this is due to the 10% pulsating exhaust gas velocity and 1% pulsating exhaust gas temperature. The results show that the impacts of the pulsating exhaust gas to the external airflows temperatures can be ignored at the 10% pulsating exhaust gas velocity and 1% pulsating exhaust gas temperature condition.

The transient exhaust gas system surface temperature and heat dissipating capacity with the periodic pulsating exhaust gas velocity and temperature are obtained and the compare the transient mean value with the steady results are presented in Fig.4 and Fig.5 in a circle (From 0.5S to 0.6S).

![Fig.4. Compare the transient mean surface temperature with the steady results.](image3)
As can be seen in Fig. 4, the transient mean surface temperatures of exhaust gas system components are higher about 40°C than the steady results. As can be seen in Fig. 5, the transient mean surface heat dissipating capacities of exhaust gas system components are higher than the steady results.

The results show that the impact of the pulsating exhaust gas to the surface temperatures and heat dissipating capacities of exhaust gas system components can be not ignored. At the 10% pulsating exhaust gas velocity and 1% pulsating exhaust gas temperature condition, the transient mean surface heat dissipating capacities of exhaust gas system components are 11.6 percent up.

4. Conclusions and Discussions

A brief summary of the work completed and important conclusions derived from this work are highlighted below.

A three-dimensional transient simulation was undertaken for the heat transfer characteristics of this vehicle exhaust system. The heat transfer characteristics with the periodic pulsating movement of the exhaust gas velocity and temperature were obtained and analyzed, and the transient mean value were compared with the steady results, and the discipline between the steady heat transfer characteristics and the periodic pulsating movement of the exhaust gas velocity and temperature were found.

At the 10% pulsating exhaust gas velocity and 1% pulsating exhaust gas temperature condition, the transient mean surface heat dissipating capacities of exhaust gas system components are 11.6 percent up.

Acknowledgments

This work was supported by the National Natural Science Foundation of China (51375168) and Science and Technology Planning Project of Guangzhou (2014J4100014)

References

[1] G. Xiao, Investigation of fluid-solid conjugate heat transfers for a Sedan exhaust system, International Workshop of Automobile, Power and Energy 6 (2011) 5076-5079.
[2] V.K Srinivasa, R.S. Srinivasa, B. Shome, Design of experiments enabled CFD approach for optimizing cooling fan performance, In SAE 2014 World Congress and Exhibition, April 8, 2014.
[3] K. Haehndel, et al., The development of exhaust surface temperature models for 3D CFD vehicle thermal management simulations Part 1 - general exhaust configurations, SAE International Journal of Passenger Cars - Mechanical Systems 6(2) 2013 1-16.
[4] N. Watanabe, M. Kubo, N. Yomoda, An 1D-3D integrating numerical simulation for engine cooling problem, SAE paper 2006, 1-1603.
[5] E. Bendell, Investigation of a coupled CFD and thermal modeling methodology for prediction of vehicle underbody temperatures, SAE paper 2005, 1-2044.
[6] E. Bendell, M. Hauenstein, Evaluation of steady and transient CFD methods for vehicle temperature prediction on the basis of a heatshield test rig, in VTMS 8 - Vehicle Thermal Management Systems Conference and Exhibition, May 20, 2007.
[7] A. Skea, A. Jolliffe, R. Harrison, et al., Integrated underbonnet thermal management simulation, VTMS 6 SAE Conference, SAE paper 2003.C599-074.
[8] P., L. Loďahl Gullberg, P. Nilsson, Cooling airflow system modeling in CFD using assumption of stationary flow, in SAE 2011 Commercial Vehicle Engineering Congress, COMVEC 2011, September 13, 2011.
[9] Z. Yang, J. Bozeman, F. Z. Shen, Thermal-velocity coupling in vehicle thermal system calculations. SAE paper 2002-01-1204.
[10] X. Zhang, M. Romzek, Computational fluid dynamics (CFD) applications in vehicle exhaust system, in 2008 World Congress, April 14-17, 2008.
[11] H. Chen, Y. Wu, Y. Hua, X. Yang, Transient analysis in vehicle thermal management and its application, Automotive Engineering 36(2) (2014) 249-253.
[12] Star CD v.3.26, Adapco Group. 2005.
[13] J. Wang, G. Xiao, Numerical simulation of the gas-solid two-phase flows in a precalciner, Journal of Wuhan University of Technology (materials science) 21(4) (2006) 177-179.