Unsteady Simulation of Conjugate Heat Transfers with Exhaust Pulsating Temperature for Vehicle Thermal Management

Guo-quan XIAO*

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

*Corresponding author

Keywords: Exhaust system, Conjugate heat transfers, Pulsating temperature.

Abstract. Unsteady simulation of heat transfers for vehicle exhaust system plays an important role in vehicle development process. Based on STAR-CD software, a sedan exhaust system model, which includes the exhaust gas, the exhaust pipes and aftertreatment devices, 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 three-dimensional unsteady simulation was undertaken for the heat transfer characteristics of vehicle exhaust system. The heat transfer characteristics with the periodic pulsating exhaust temperature were obtained and analyzed, and the unsteady mean value were compared with the steady results, and the discipline between the steady heat transfer characteristics and the periodic pulsating exhaust temperature were found.

Introduction

Unsteady simulation of heat transfers for vehicle exhaust system has become an important part of the vehicle development process. About 1/3 of the total chemical energy that enters an engine as fuel is converted to useful crankshaft work, and about another 1/3 of the total energy that must be dissipated to the surroundings by some mode of heat transfer and friction. This leaves 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.

It is necessary to use multi-dimensional computational dynamic fluid (CFD) codes to estimate components 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 vehicle temperatures remains a largely test-driven process. Several examples of previous studies may be found in the literature, using CFD packages to compute multi-mode heat transfer in vehicles [2-7]. The most detailed is due to Bendell, E. where a coupled steady-state CFD and thermal study was undertaken at full-vehicle scale [8-9]. 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 [10]. 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 component 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 [11-14]. CHEN Hong-Ming presented a unsteady 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 code, the feasibility of unsteady thermal analysis is verified by test [15].

The paper is organized as follows: the next section describes the computational model. The following two sections present numerical methods and boundary conditions respectively. The
Conjugate heat transfers for vehicle exhaust system are described in the results section. Summary are given in the final section of the paper.

**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 200 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. 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 domain used is shown in Figure 1.

![Car model and computational domain](image)

Figure 1. The car model and computational domain used in the simulation.

The sedan component surfaces were discretized with triangle meshes with Hypermesh, Surface mesh quality is modified by controlling skewness 0.85. The surface mesh used is shown in Figure 2.

![Surface mesh](image)

Figure 2. The surface mesh used in the simulation.

Tetrahedral cells was used to discreted the fluid and the solid in the computational domain with TGrid, and the volume mesh quality is modified by controlling mesh skewness 0.95, there are about 3.3 million cells used is shown in table 1.

| Table 1. The grid distribution used in the simulation[unit: thousand]. |
|------------------|------------------|------------------|------------------|------------------|------------------|
| type             | Fan              | Heat exchangers  | Exhaust gas      | Pipe wall        | External airflow | Total            |
|                  | MRF              | porousmedium     | fluid            | solid            | fluid            |                  |
| Grid             | 300              | 70               | 100              | 80               | 2,850            | 3,300            |
Numerical Methods
The conjugate heat transfer simulations were carried out by using STAR-CD [16]. Recent advances in computer speed and power, coupled with STAR-CD's High-Reynolds formulation of the k-epsilon turbulence model with standard wall function and the DO model for the radiation heat transfer makes it feasible to integrate these three major vehicle elements into a single simulation, as was done in this example.

The condenser and radiator flow resistances were modeled as porous media in the normal way in STAR-CD from manufacturer data. The heat transfer to the condenser and radiator 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. In the solid phase only the energy equation would be solved.

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.

Boundary Conditions
Since the exhaust gas in study is intrinsically unsteady, the unsteady heat transfer characteristics of exhaust system were simulated based on the steady results at vehicle uphill condition with full load, and the periodic pulsating exhaust gas temperature were given as follows Eq.1:

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

(1)

Where, \( T_{in} \) is unsteady temperature, \( T_s \) is mean temperature, \( T_{max} \) is maximal pulsating temperature, \( \omega \) is pulsating frequency.

The periodic pulsating values of the exhaust gas temperature were coded into subroutine UFILE [16], which can be called by STAR-CD software, and the time step \( \Delta t \) was given by the engine speed.

The other boundary conditions such as ambient temperature, exhaust gas specific heat, emissivity and the vehicle velocity, radiator power, condenser power and fan speed be accurately represented by the steady simulation at vehicle uphill condition with full load.

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

The unsteady airflows temperature with the periodic pulsating exhaust gas temperature are presented in Figure 3 \((Y=80\text{mm})\) and Figure 4 \((Z=150\text{mm})\) and analyzed quality in a circle(From 0.5S to 0.6S).

![Figure 3. Temperature of y=80mm plane in a circle.](image)
As can be seen in Figure 3 and Figure 4, the predicted external airflows pulsating temperatures are lower than 10°C. The results show that the impact of the pulsating exhaust gas to the external airflows temperatures can be ignored at the 1% pulsating exhaust gas temperature condition.

The unsteady exhaust gas system surface temperature and heat dissipating capacity with the periodic pulsating exhaust gas temperature are obtained and the compare the unsteady mean value with the steady results are presented in Figure 5 and Figure 6 in a circle (From 0.5S to 0.6S).

As can be seen in Figure 5, the unsteady mean surface temperatures of exhaust gas system components are higher about 40°C than the steady results. As can be seen in Figure 6, the unsteady 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 temperature condition, the unsteady mean surface heat dissipating capacities of exhaust gas system components are 10.6 percent up. And the discipline between the fluid-solid conjugate heat transfer characteristics and the periodic pulsating movement of the exhaust gas velocity were found.

**Summary**

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

A sedan exhaust system model, which includes the exhaust gas, the exhaust pipes and after treatment devices, 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 three-dimensional unsteady simulation was undertaken for the heat transfer characteristics of sedan exhaust system. The results with the periodic pulsating exhaust gas temperature were obtained and analyzed, and the unsteady mean value were compared with the steady results, and At the 10% pulsating exhaust gas temperature condition, the unsteady mean surface heat dissipating capacities of exhaust gas system components are 10.6 percent up.
Simulations such as these help designers shorten the time to market by reducing the number of prototypes built for testing.

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

References