

The Development of Exhaust Surface Temperature Models for 3D CFD Vehicle Thermal Management Simulations Part 1 - General Exhaust Configurations

Kristian Haehndel
BMW Group - RMIT University

Torsten Frank, Frieder Martin Christel, Carsten Spengler and Gerrit Suck
BMW Group

Sylvester Abanteriba
RMIT University

ABSTRACT

The thermal prediction of a vehicle under-body environment is of high importance in the design, optimization and management of vehicle power systems. Within the pre-development phase of a vehicle's production process, it is important to understand and determine regions of high thermally induced stress within critical under-body components. Therefore allowing engineers to modify the design or alter component material characteristics before the manufacture of hardware. As the exhaust system is one of the primary heat sources in a vehicle's under-body environment, it is vital to predict the thermal fluctuation of surface temperatures along corresponding exhaust components in order to achieve the correct thermal representation of the overall under-body heat transfer.

This paper explores a new method for achieving higher accuracy exhaust surface temperature predictions. To avoid the experimental dependency of fixed exhaust temperature surfaces, a 1-Dimensional fluid stream was integrated within a 3-Dimensional exhaust surface piping network. To encompass the 3-Dimensional effects of combusted internal gas flow, correctional factors were employed. These correctional factors were derived from a study of exhaust gas dynamics and the individual influences on conjugate heat transfer within internal exhaust networks. Four primary phenomenas were found to have significant influence on internal heat transfer coefficients; hence the combination of these effects could be described through a total augmentation factor.

Several exhaust configurations were simulated using an in-house heat transfer prediction tool which utilised the work presented in this paper. The results of two particular exhaust configurations are presented within this paper. It was found that both exhaust configurations achieved a good trend in comparison to experimentally derived data. The regions of poor accuracy were found to predominantly exist in components with an internal structure. Over and under predictions of surface temperature were not only dependent on the exhaust model but also on the mass flow rate experienced within the piping network. The source of the error was found to be the multi-layered arrangement assigned to internally structured components within the simulation method of 3-Dimensional surface shells.

CITATION: Haehndel, K., Frank, T., Christel, F., Spengler, C. et al., "The Development of Exhaust Surface Temperature Models for 3D CFD Vehicle Thermal Management Simulations Part 1 - General Exhaust Configurations," *SAE Int. J. Passeng. Cars - Mech. Syst.* 6(2):2013, doi:10.4271/2013-01-0879.

INTRODUCTION

The ability to computationally predict thermal fluctuation and temperature induced stress avoids the costly and time consuming prototype testing in the validation process. In the pre-development stage of a vehicle (prior to prototype

construction), engineers utilise numerical approximations to evaluate adverse areas prior to 'freezing' of the design. Furthermore, thermal prediction is also employed for validation of known problematic areas.

During component testing it is common practice that a thermocouple is positioned in the region of the most extreme

temperature predicted from either physical or computational experience. This is a safe-guard process, whereby engineers have probed highest experienced temperature data ("Hot-Spots") for the re-design or modification of vital components when necessary. Not only is the position of a sensor temperature dependent, it is also dependent on the known (previously experienced) thermal stress on surrounding components.

For many years engineers and researches have had difficulty predicting the surface temperatures of components within the exhaust system. Numerous journals and papers have been presented on the topic however many areas of exhaust heat transfer phenomena are still poorly understood (Depcik & Assanis, 2002). As each different engine produces a specific gas flow characteristic dependent on valve design, timing events and exhaust configurations, it is challenging to develop a single mathematical correlation to describe the heat transfer phenomenon experienced within a pipe or component. Additionally the surrounding environments (both ambient conditions and the vehicle under-body) alter the external convection and radiation properties experienced by the outer surface of the exhaust system. Such factors significantly affect the correlation between experimentally derived test data and that obtained from numerical simulations, and has the potential to be a large source of error. Hence the minimisation of these errors is of a primary importance to the heat management division of the BMW Group.

CONTRIBUTING FACTORS TO EXHAUST HEAT TRANSFER

The Reynolds Number (Re) greatly affects the transfer of heat from the exhaust gas to surrounding components and is used to describe the flow regime using an averaged classification. Equation 1 represents the standard Reynolds equation which can be applied to an exhaust system.

The entrance effect of the exhaust flow is affected by a combination of the velocity and thermal boundary layers. The non-slip condition on the surface of the pipe causes a parabolic velocity profile of the exhaust gas. The temperature difference between the exhaust gas and the pipe wall, along with the non-slip condition, results in a variation of the corresponding thermal profile (Cengel & Turner, 2005).

The Prandtl Number (Pr) represents the relative thickness of the velocity to thermal boundary layers [Eqn. 2]. It is a primary factor used in calculating the Nusselt Number (Nu), a representation of the heat transfer from the combusted gas to the exhaust component walls. In general the Nusselt number can be described as the power combination of two variables, the Reynolds number and Prandtl number [Eqn. 3]. Many technical papers have been written describing their own individual derivations of this number, however most correlations are empirically dependent and only applicable to identical simulation scenarios (Depcik & Assanis, 2002). Hence the well known Sieder-Tate model [Eqn. 4] with its

extensive empirical validation was utilised as a base Nusselt Number formulation for this investigation.

$$Re = \frac{\text{INERTIAL FORCES}}{\text{VISCOUS FORCES}} = \frac{V_m D}{\nu} = \frac{\rho V_m D}{\mu} \quad (1)$$

$$Pr = \frac{\text{Molecular diffusivity of momentum}}{\text{Molecular diffusivity of heat}} = \frac{\nu}{\alpha} = \frac{\mu C_p}{k} \quad (2)$$

$$Nu = a Re^b Pr^c \quad (3)$$

$$Nu = 0.027 Re^{0.8} Pr^{1/3} \left[\frac{\mu_{bulk}}{\mu_{skin}} \right]^{0.14} \quad (4)$$

COMMON INDUSTRIAL METHOD

The former practices employed within the BMW Group to computationally establish under-body heat transfer involved fixing the temperature of exhaust components from empirically derived data. This method allowed for the conjugate heat transfer calculation to be performed. However this process inherently contained several disadvantages which fail to establish the objectives of satisfactory heat management. This included:

- The dependency of known surface temperature data, either derived from experimental testing or professional experience.
- Inaccurate representation of the temperature distribution along the surface of a component.
- Inaccurate calculation of conjugate heat transfer.

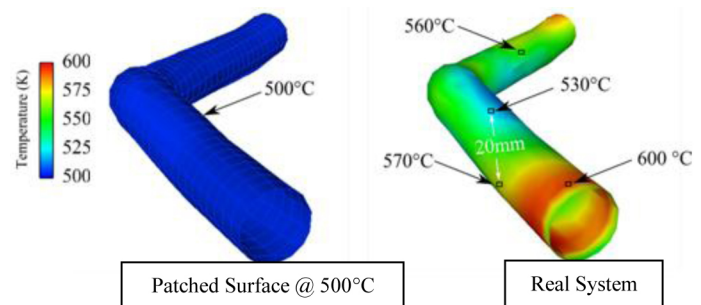


Figure 1. Comparison between temperature fix and heat transfer coefficient integration.

Many components experience a temperature variation which influences the entire under-body heat transfer process. The temperature fixing of a surface, results in the radiation being uniformly exposed with a constant magnitude to surrounding components. It is evident within Figure 1 that the temperature difference along the surface of a component can vary significantly with position. This variation influences the magnitude of emitted radiation (as radiation has a power dependence (T^4) on temperature) and has a direct

consequence on the heat transfer rate on surrounding under body parts.

PROPOSED METHOD

The proposed method introduces a 1-*Dimensional* fluid stream within the exhaust system (Figure 2), where the combination of improved internal heat transfer coefficients and alternative modelling techniques allow for the integration of 3-*Dimensional* gas dynamic influences. The inlet conditions of the combusted gas (temperature and mass flow rate) are derived from experimental measurements.

The thermal influence of the turbine work (heat extraction) and the chemical kinetics of the CAT (heat addition) are considered. The energy manipulations of these processes are implemented within the proposed methodology through imposed heat functions. The models used to generate the appropriate heat values were not the focus of this investigation. However they will be presented in future publications.

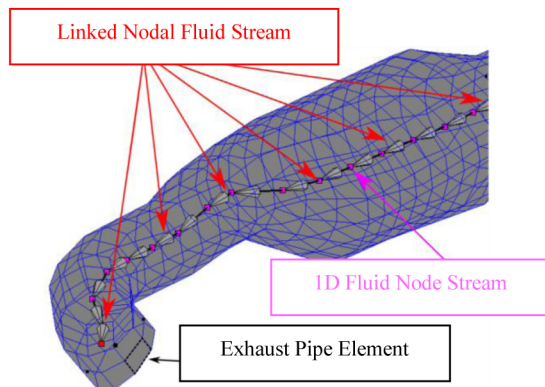


Figure 2. Internal fluid stream within exhaust component example (Adapted from ThermoAnalytics, 2009).

This method provides a step forward towards the independence of experimental data, being used within a computational environment.

The advantages include:

- Improvement of overall simulation quality and thermal resolution within the exhaust system.
- The development of higher accuracy computational models.
- Reduces the excessive computational expense of full 3D exhaust modelling.

Convection Augmentation Factor

Due to the complexities of multi-variable influences on exhaust heat transfer, a majority of literature presented Nusselt number correlations that are constructed from experimentally specific testing conditions (Kandylas & Stamatelos, 1999). These models become limited in their application to alternative exhaust configurations or variable

engine characteristics. In industry it is common practice to use a convective augmentation factor (CAF) to compensate for an inaccurate heat transfer simulation of flow phenomena not incorporated within a theoretical Nu model. This method avoids the dependency on empirically derived correlations, whereby the same methods for Nusselt number validation are not universally employed amongst the vast set of research authors. The investigation searched to understand the augmentation influence of each phenomenon based on its individual CAF value with standard theoretical Nusselt relations for straight pipes. The CAF can be described as the ratio of the $Nu_{\text{Experienced}}$ within the real environment to the theoretical (Nu_{Ideal}) experienced within the computational domain. Hence the Nu_{Real} can be described below in Equation. 5.

$$Nu_{\text{Real}} = Nu_{\text{Ideal}} * CAF \quad (5)$$

Utilising Sieder-Tate model as an initial base Nu number correlation the formulation becomes:

$$Nu_{\text{Real}} = 0.027Re^{0.8}Pr^{1/3} * \frac{Nu_{\text{Experienced}}}{Nu_{\text{Ideal}}} \quad (6)$$

It can be seen that the Nu_{Real} [Eqn. 6] is simply the mathematical enhancement of the Nu_{ideal} to match the experienced experimental heat transfer within a component. The constitution of the CAF for each exhaust component is assumed to be the combination of the four phenomena discussed in the next section.

NUMERICAL MODELING AND TEST SET UP

The following section aims to provide the general tools utilised to investigate and improve exhaust surface temperature models. The process involves the application of numerical methods to simulate exhaust system heat transfer with the combination of theoretical modelling and experimental validation.

Numerical Modeling

Many commercially available software packages exist, which accurately predict aerodynamic and heat transfer characteristics of vehicle components and sub-assemblies. The main software platform to be utilised within this investigation is *Radtherm*. This software is used to calculate the conduction and radiation values for the total simulation. It is additionally used to implement the 1-*Dimensional* correlations (through fluid streams) through the exhaust system. The under-body airflow (convection) is simulated using *Star-CCM+*. The combinations of these software packages permit the entire under-body heat transfer analysis.

The coupling process proceeds with the following 5 steps:

1. An initial steady state complete vehicle CFD simulation is conducted within *Star-CCM+*.

2. Convective parameters (h , T_{Fluid}) are then imported into *Radtherm* and mapped on to the corresponding components.

3. The thermal simulation in *Radtherm* proceeds by utilising the previous mapped values for the conjugate heat transfer calculation. Additionally the fluid streams through the exhaust system with the proposed 1-Dimensional correlations are actively participating in the thermal calculation.

4. The temperatures (T_{Wall}) of corresponding under-body component surfaces are then exported to update the CFD solution.

5. This coupling process continues, cycling data between thermal and fluid models until convergence is achieved (as shown in Figure 3).

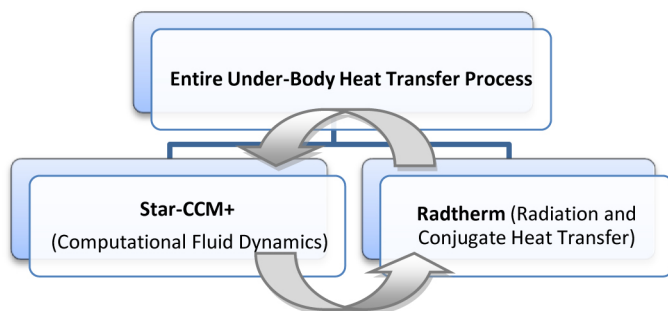


Figure 3. Coupling Process

Experimental Testing

Validation of numerical results is performed by measurement of the corresponding parameters during road testing and/or climatic wind tunnel testing of the vehicle. Validation additionally included fixed engine test bench experiments, whereby the “hot end” (manifold to post-cat components) was temperature tested. Engine bay convection is artificially produced (via a external fan placed in front of the test bench) through matching the average on road forced convection to a standard fan speed.

A factor influencing the positioning of thermocouples is the effect probes inflict on external fluid dynamics. As a result the positioning of probes is model specific. Having large amount of probes incrementally along the exhaust pipe tends to influence the heat transfer coefficients induced by external under body convection. To avoid this effect probes are placed in regions of interest and critical relevance.

The thermal variation experienced on a specific component such as take-down pipe (shown in Figure 4) can be of significant interest in the validation of integrated 1-Dimensional models. Hence a thermal (infra-red) camera is used (during the experiment) as a means of confirming the temperature distributions achieved within the corresponding numerical models.

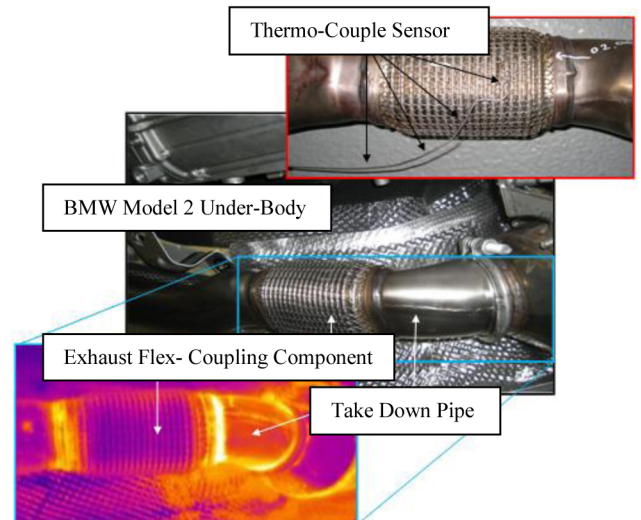


Figure 4. Thermal Couple Positioning

THE STUDY OF EXHAUST GAS DYNAMICS AND THE INFLUENCE ON HEAT TRANSFER

A survey of the open literature identified four primary exhaust system heat transfer phenomena as the main influences on thermal behaviour: entrance effects, engine induced pulsation, surface condition and geometry. The conditions of these four phenomena constitute the basis of this paper.

Entrance Effects

It is logical that the boundary layer thickness grows with increasing distance along a tube; until it reaches 10%-20% of the pipes radius due to the effects of viscosity. The distance to this point is known as the velocity/thermodynamic entrance length and the fluid characteristics within this region are classified as “developing”, as the maximum temperature or respective velocity is yet to be reached. Understanding this boundary layer development can assist engineers in the selection of heat transfer correlations which encompass the phenomena.

Farrugia et al. (2006) investigated the influence of entrance phenomena on the entire heat transfer process within straight pipe extension from an exhaust port of a 1.9 litre, four cylinder 1992 Saturn spark-ignition test bench engine. Using cycled average heat flux measurements he was able to adapt the Boelter et al. (1948) entrance formulation to derive the following CAF correlation [Eqn. 7]. The CAF model aims to compensate a base heat transfer correlation (Sieder-Tate) only for the effects of entrance, hence enabling engineers to distinguish between possible phenomena influence. These effects can be described purely as a function of distance (x) and pipe diameter (D), indicating that the entrance augmentation is dominated by geometrical constraints and