

Experimental Characterisation of Heat Transfer in Exhaust Pipe Sections

C. D. Bannister, C. J. Brace and G. D. Lock
University of Bath

J. Taylor, T. Brooks and N. Fraser
Mahle Powertrain

Copyright © 2008 SAE International

ABSTRACT

This paper describes the characterisation of heat transfer in a series of 11 test sections designed to represent a range of configurations seen in production exhaust systems, which is part of a larger activity aimed at the accurate modeling of heat transfer and subsequent catalyst light off in production exhaust systems comprised of similar geometries. These sections include variations in wall thickness, diameter, bend angle and radius. For each section a range of transient and steady state tests were performed on a dynamic test cell using a port injected gasoline engine. In each case a correlation between observed Reynolds number (Re) and Nusselt number (Nu) was developed. A model of the system was implemented in Matlab/Simulink in which each pipe element was split into 25 sub-elements by dividing the pipe into five both axially and radially. The modeling approach was validated using the experimental data.

The steady state relationship between Re and Nu allow heat transfer in the test section to be predicted with acceptable accuracy over transient test cycles and demonstrates good agreement with relationships in the literature.

The model accuracy was enhanced by developing empirical models of heat transfer during the warm up stage of the transient test.

INTRODUCTION

Increasingly stringent gasoline emissions legislation, in Europe and across the world, has been a driving force behind engine calibration optimisation, advancements in catalyst technology and the reduction of catalyst light-off times. The development of new engine and vehicle platforms has become progressively more expensive as consumer expectations and engine complexity increase. For this reason, manufactures and automotive consultants are relying more heavily on simulations to

reduce development times, rather than expensive experimental programs.

Mahle Powertrain (MPT) contracted the University of Bath to develop a Matlab-Simulink based model which could accurately predict gas temperatures at the front face of the catalyst based on port temperatures obtained, either experimentally, or by means of an engine model run in the GT-Power software package. Predicted gas temperatures could be used to estimate the effects of changes in engine calibration or exhaust system geometry (such as catalyst positioning) on catalyst light-off times.

The starting point of the project was an initial model which had been created by Mahle Powertrain based on fundamental heat transfer calculations outlined by Shayler et al. [1] which treated the sections as lumped masses and temperatures as well as assuming quasi-steady flow conditions within the pipe. The pipe section itself is treated on the basis of turbulent convective heat transfer from the gas to the pipe inner wall, conductive transfer radially through the wall and axially along the section and by natural convection to the surroundings from the pipe outer wall. An energy balance is then struck between the energy loss from the gas flow and energy dissipated to the element's surrounding via conduction, convection and radiation.

The model also utilised the 'resistor analogy' discussed and presented by Eriksson [2] whereby the modes of heat transfer are considered as resistors positioned in series or in parallel with the sum of the reciprocals equating to the total heat transfer coefficient as given in Equation 1.

$$\frac{1}{h_{tot}} = \frac{A_i}{A_o} \frac{1}{h_{cv,i}} + \frac{1}{h_{cd}} + \frac{1}{h_{cv,e} + h_{cd,e} + h_{rad}}$$

Equation 1 – Total heat transfer coefficient calculated using the 'resistor analogy' [2]

This study examined opportunities to develop this model, improving its accuracy and increasing its usefulness as a predictive development tool.

MODEL CHANGES AND DEVELOPMENT

THICK-WALLED SECTIONS AND FLANGES - For thin walled metal sections, it is often acceptable to assume that the inside wall temperature is equal to the temperature on the outside however, for thicker-walled sections, flanges and materials with lower thermal conductivity this assumption is incorrect. Depending on the mass of the section a considerable amount of energy may be required to raise the section wall temperature with significant temperature gradients present in both axial and radial directions.

Using an example of a flange (Figure 1), Konstantinidis *et al* [3] predicted the unsteady heat flow by the application of the finite difference method.

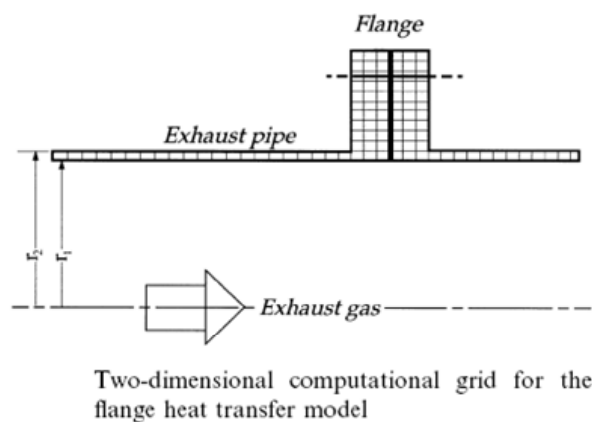


Figure 1 – Konstantinidis *et al* [3] flange representation

Based on the methodology outlined by Konstantinidis *et al* [3] and Buchner *et al* [4] a new model was developed by the University of Bath to solve non-steady conduction within flanges and thick walled sections.

The principle involved splitting the flange into a number of equal axial and radial subsections with conduction equations solved for each section based on the condition of the surrounding sections. Initially the model was set up to split a given flange into 5 axial and radial elements or equal thickness and length. Using the example of an exhaust manifold flange, Figure 2 shows the energy flow into and out of the flange.

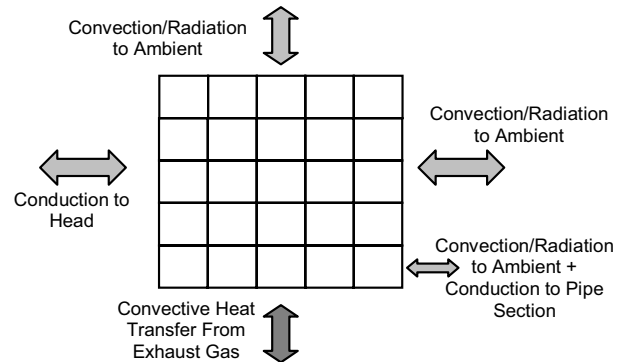


Figure 2 - Exhaust manifold flange energy flow pathways

The 5 by 5 grid represents the pseudo finite element approach adopted to represent the conduction within the flange section non-steady heat flow between the nodes. In addition, the energy flows associated with conduction to the head and adjacent pipe section, radiation and convection the surroundings on the outer flange faces and convective heat transfer between the gas and inner wall of the flange are included and solved via an energy balance including an energy storage term.

Figure 3 shows a portion of the flange section model created in the Simulink environment. Nodes are colored dark gray with white blocks connecting them representing conduction between nodes or adjacent pipe sections. Forced convection between the gas and wall is modelled within the light grey blocks.

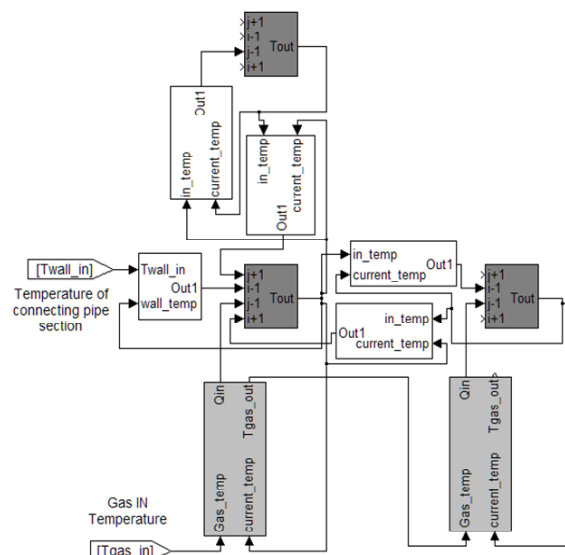


Figure 3 - Flange section Simulink model

ADOPTION OF THE 'FLANGE MODEL' FOR ALL SECTIONS - Although thin wall sections do not necessarily require the consideration of non-steady conduction the resultant penalty in computational running time is minimal. The use of the non-steady conduction model for all pipe sections offers the added benefit of reducing the complexity of the initial model set-up with only one 'block' being required to describe all pipe thicknesses. For these reasons it was decided to adopt the quasi-finite element 'flange model' approach for all pipe sections regardless of wall thickness. This structure forms the basis of the blocks used in the model construction discussed later.

EXPERIMENTAL MEASUREMENT OF HEAT TRANSFER

The heat transfer calculations contained within the Matlab-Simulink model were originally obtained from published literature however, what became apparent during the review of these published sources was the variations between relationships from different authors varied significantly depending on the system being modelled.

The common way of describing the convective heat transfer relationship between the gas and inner wall of the pipe, of a given system, is by examining the correlation between the Reynolds number (Re) of the gas and the Nusselt number (Nu). The Nusselt number is a dimensionless number used to measure the enhancement of heat transfer due to convection and can be expressed for a simple pipe section as:

$$Nu = \frac{h_{cv,i} D_i}{k_f}$$

Equation 2 – Nusselt Number Relationship

where $h_{cv,i}$ is the internal convective heat transfer coefficient, D_i is the internal diameter of the pipe section and k_f is the thermal conductivity of the fluid.

As the Nusselt number is a measure of the increased heat transfer due to convection it follows that its value will be related to the state of the fluid flow to which the surface is exposed and thus, the Reynolds number. The general form of the Re/Nu relationship is given in Equation 3 and some published correlations are shown in Table 1.

$$Nu = c_0 Re^{c_1} Pr^{c_2} \left(\frac{\mu_{bulk}}{\mu_{skin}} \right)^{c_3}$$

Equation 3 - General form of Re/Nu Relationship

Author	C0	C1	C2	C3
Sieder-Tate [5]	0.027	0.8	1/3	0.14
DOHC [6]	0.26	0.6	0	0
Valencia [6]	0.83	0.46	0	0
Malchow <i>et al</i> [7]	0.0483	0.783	0	0
Wendland [8]	0.081	0.8	1/3	0.14

Table 1 - Published Re/Nu Relationships

When the relationships outlined in Table 1 are applied to a simple straight pipe section, using measured gas temperatures as an input, variations in predicted gas outlet temperatures can be observed (Figure 4). All methods are seen to underestimate the heat transfer during the initial stages of the arbitrary drive cycle and overestimate during the latter stages when compared with measured outlet temperatures.

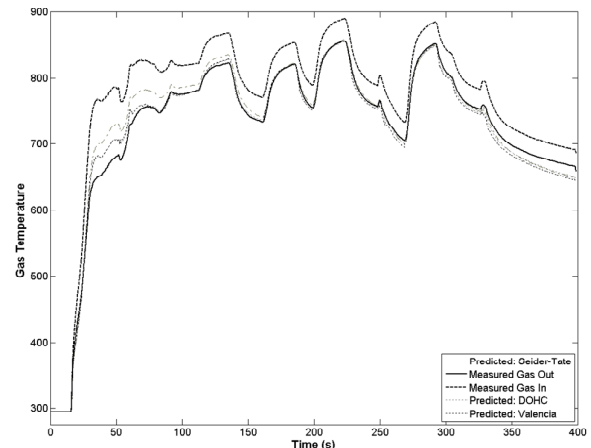


**Figure 4 - Variations in Re/Nu Relationships from Different Authors
(Gas temperatures measured in °K)**

Due to the variations observed in the predicted results obtained using published relationships this study concentrated on developing a generic model which modifies the predicted Re/Nu relationship based on system geometry. In order to achieve this it was necessary to characterise the Re/Nu relationships experimentally for pipe sections of varying geometry.