

Transient Modelling of Vehicle Exhaust Surface Temperature

Alaa El-Sharkawy
FCA US LLC

Amr Sami and Abd El-Rahman Hekal
Optumatics LLC

Dipan Arora and Masuma Khandaker
FCA US LLC

ABSTRACT

In this paper, the development of a transient thermal analysis model for the exhaust system is presented. Given the exhaust gas temperature out of the engine, a software tool has been developed to predict changes in exhaust gas temperature and exhaust surface temperature under various operating conditions. The software is a thermal solver that will predict exhaust gas and wall surface temperatures by modeling all heat transfer paths in the exhaust system which includes multi-dimensional conduction, internal forced/natural convection, external forced/natural convection, and radiation. The analysis approach involves the breaking down of the thermal system into multiple components, which include the exhaust system (manifold, takedown pipe, tailpipe, etc.), catalytic converter, DPF (diesel particulate filter), if they exist, thermal shields, etc. All components are modeled as 1D porous and 1D non-porous flow streams with 3D wall layers (solid and air gaps). The aggregate of all these components is referred to as the thermal network. All relevant modes of heat transfer between the parts, i.e., forced convection, conduction and radiation are considered. Analysis of the thermal network is conducted by coupling all the components in the form of a computational grid, which for the most part will be conducted via implicit numerical procedure. This will make the solution robust, particularly for conduction dominant conditions. Results from the transient analysis are compared to actual vehicle tests under various test conditions

CITATION: El-Sharkawy, A., Sami, A., Hekal, A., Arora, D. et al., "Transient Modelling of Vehicle Exhaust Surface Temperature," *SAE Int. J. Mater. Manf.* 9(2):2016, doi:10.4271/2016-01-0280.

INTRODUCTION

During this work, a software analysis, Virtual Time-Temperature Analysis (VTTA) is developed for estimation of exhausts surface temperature during transient driving conditions. The need for this type of simulation is critical for the vehicle thermal development step. Conducting thermal analysis early in the program will help identify areas that need thermal protection or material upgrade. CFD analysis tools are normally used to predict component temperature during steady state conditions. The level of CFD analysis can be relatively accurate; however, it is too costly to run a full vehicle transient CFD analysis. Therefore, the purpose of this work is to develop transient thermal analysis model to predict exhaust gas temperature and exhaust surface temperature under various driving conditions. Results for transient exhaust surface temperatures at various locations along the exhaust system are used in a separate study, by the authors, to estimate the transient component temperatures. Exhaust surface temperatures can be used also as input for CFD analysis. The exhaust system can be described in a geometry file. The analysis can therefore predict the effect of changes in exhaust system configuration, heat shield design and changes in the exhaust gas temperature. The main input therefore, includes exhaust gas temperature out of the engine at various loads,

exhaust system configuration, radiator out air flow and air temperature and under body airflow. Results from this analysis will focus mainly on the exhaust gas temperature and exhaust surface temperatures during a variety of vehicle driving conditions which can include light load, high load conditions as well as slow speed and high speed driving conditions.

GEOMETRY PROCESSING

The simulation requires a work-flow consisting of geometry data preprocessing, case data preprocessing, solving the problem and post-processing for data analysis. The software is console based so all stages are interacted with directly from the console window. The geometry preprocessor is invoked by the user after executing the VTTA software. The geometry preprocessor will read in a geometry file, process it, generate a computational grid, and save it to a file that will be used later by the VTTA solver. For a given geometry file the user will only need to invoke this step once, which will reduce the run time in the scenario of multiple cases for a single geometry.

EXTERNAL FLOW MODEL

To calculate the external heat transfer coefficients using the previous correlations it is necessary to obtain the external flow conditions at all exterior surfaces. The analysis approach can use some of the following methods:

Method 1

Solve the momentum and continuity equation with a simplified computational domain for the under hood and under body regions.

Method 2

Use the cooling flow input as an estimate to external velocity of the exhaust and components. Depending on the location of the part either the under-hood cooling flow or the under body flow is applied. Also depending on the exterior surface orientation of each part a velocity corrector is applied to account for blockage effects and to decide whether the flow is parallel or cross.

After testing both methods it was decided to drop method1 and use method2 for the following reasons:

- The angle between the face normal and the main flow direction is used to decide whether the flow is moving across the face or parallel to it
- A velocity correction factor was devised to account for the low velocity at backward facing faces
- A velocity correction factor was devised to account for the lower velocity at faces that are facing upward and near the vehicle floor

It was found that it will not be feasible to use method1 in the VTTA tool as very small time steps were required to achieve a numerically stable solution (1000 times smaller than what is needed by the thermal solution). Although much more accurate, method1 was found to be computationally very expensive. After testing the VTTA tool with method2 it was found that it can reasonably capture the main features of the expected exterior surface behavior (e.g. difference between top and bottom surfaces) with minimal computational cost. Thus method2 was selected given its acceptable accuracy, simplicity, stability problems faced with method1, and the computational time constraints of the VTTA tool.

CASE PREPROCESSOR

The case preprocessor is invoked after reading the grid file successfully. The case preprocessor will read in a case specific file where operating/ boundary condition data, and output settings will be provided by the user.

VTTA Case File

The case file comprises of four main sections: a solution time specification, a monitor points specification, and an operating/ boundary conditions specification. In the solution time specification it is required to specify n number of intervals where for each interval a time step and end time is given. The user will also specify an output

time step for solution files writing frequency in this section. For the monitor points specification it is required to specify n number of monitoring points where for each point a segment ID, segment length fraction, and segment layer ID (specify 0 for flow monitoring) and pole ID (top or bottom) is required. In the operating conditions specification it is required to define for each segment an initial temperature, a top and bottom external air temperature, and a top and bottom enclosure radiation temperature. To control the internal and external convection rate obtained from built in correlations in the VTTA solver, an optional multiplier is read for all segments. Finally In the boundary conditions specification section a time table with exhaust mass flow rate, exhaust gas temperature and external air velocity is to be read to update the boundary conditions with time.

INPUT FILES SPECIFICATIONS

Geometry input files consist of the following components

- Exhaust pipe and its segments
 - Type (e.g. straights, bends, splits, cat converter, muffler, etc.)
 - Shape
 - Cross sectional area parameters
 - Length
 - Number of wall layers/gaps
 - Wall layer/gap thicknesses
 - Wall layer/gap widths
 - Wall layer/gap materials
 - Connections
- Under-hood/Underbody flow segments
 - Shape
 - Cross sectional area parameters
 - Length
 - Connections

The geometry file is a text file that consists of a material input section and an exhaust segments geometry input section. In the material input section the user will either select a material from the software library or provide properties for a new user defined material. The exhaust segments input section comprises of several single line inputs for each segment based on its type and number of sub elements (wall layers, enclosures, ports).

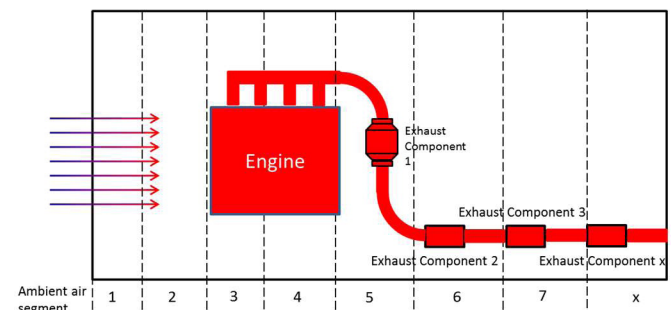


Figure 1. Schematic diagram for the under-hood and underbody flow model

There are four main element types that can be used by the user to build the exhaust path; straight pipes, porous pipes, bends, and splits. The user can divide the exhaust flow path into a series of segments formed with the four main elements based on the changing geometry of the path. Complex geometries like catalytic converters and mufflers are broken down into basic elements to model their thermal inertia and flow behavior more accurately. For example the catalytic converter should be divided into an expanding cone, a porous pipe, a straight pipe, a porous pipe again, and then a contracting cone. Porous pipes should be used to model segments like the catalytic converter brick segment to account for the enhanced heat transfer due to the increased surface area and small diameters of the flow elements. The straight pipe input comprises of three main parts; a flow path geometry specification, a pipe wall layers specification, and external pipe enclosure specification. The flow path is mainly defined by its inlet diameter, outlet diameter, and length.

The user can define expanding and contracting cones with the pipe element by giving different diameters for the inlet and outlet. The wall layers input is used to define the pipe wall thicknesses and layering (air gaps or different materials), and span angle. Each pipe wall layer is broken down internally into a top and bottom segment. This is useful where for example heat shields cover a certain span of a pipe. The pipe external enclosures input are used to define external radiation interaction with the pipe surface and to define the external convection heat transfer mode. Several enclosures segments with different geometries and emissivities can be defined. Gaps in the span between enclosure surfaces are defined as an ambient enclosure internally.

Bends and porous pipes have identical inputs as straight pipes, however the former requires a bend angle input and the latter requires a cell density and porosity input. Unlike pipes and bends more than two ports can be defined for the split segment input, with the rest of the input is identical to the straight pipe segment. Figure 2 and Figure 3 show discretization of the exhaust system for solution of energy equation.

THERMAL ANALYSIS SOLVER

The VTTA solver is invoked and will begin the solution after the case preprocessor successfully processes the input file data.

Enthalpy Equation Discretization

The energy equation is shown below in the form of enthalpy balance to account for changes in exhaust gas temperature and thermal properties.

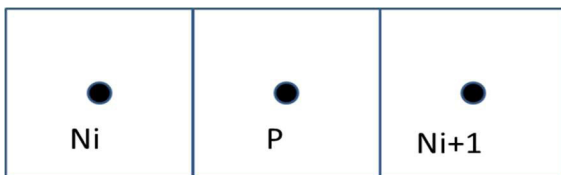


Figure 2. Discretization of the exhaust system for solution of energy equation

$$\frac{\delta \rho H}{\delta t} + \vec{\nabla} \cdot \rho u H = \frac{\delta p}{\delta t} + \vec{u} \cdot \vec{\nabla} p + \vec{\nabla} \cdot \frac{k}{c_p} \vec{\nabla} H + \dots \quad (1)$$

$$\left(A \frac{k}{c_p} \vec{\nabla} H \right)_j^{n+1} = A_f h_{eff} * \left[\left(\frac{H}{c_p} \right)_p - \left(\frac{H}{c_p} \right)_N \right] \quad (2)$$

$$h_{eff} = \frac{h_p h_n}{h_p + h_n}, h_{p,n} = \frac{k}{dx} \text{ (for solid)} \\ h_{flow} \text{ (for fluid)}$$

$$\frac{[(\rho H)_p V_P]^{n+1} - [(\rho H)_p V_P]^n}{\Delta t} + \sum_{facej} C_j^{n+1} \\ = \sum_{facej} \left(A \frac{k}{c_p} \vec{\nabla} H \right)_j^{n+1} + S_u \quad (3)$$

The convective flux term across a face is modeled using the upwind differencing scheme, where for each face:

$$C_i^{n+1} = \begin{cases} -\dot{m}_{face} H_p, & \dot{m}_{face} \text{ is outflowing} \\ \dot{m}_{face} H_N, & \dot{m}_{face} \text{ is inflowing} \end{cases} \quad (4)$$

And the source term

$$S_u = \dot{Q}_{rad} + \dot{Q}_{user}$$

Where,

$\dot{Q}_{rad}$ is the radiation heat flux obtained by separately solved radiosity matrix

$\dot{Q}_{user}$ is an optional heat generation term set by the user (e.g. to model the heat generation in the DOC)

Radiosity Equation

Radiosity is the radiant flux leaving (emitted, reflected and transmitted by) a surface per unit area.

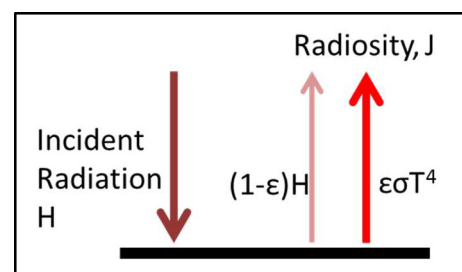


Figure 4. Radiation of effect of radiation energy on a surface