

The Development of Turbine Volute Surface Temperature Models for 3D CFD Vehicle Thermal Management Simulations: Part 3: Exhaust Radial Turbine Volute Systems

Steve De Vos and Kristian Haehndel
BMW Group - RMIT University

Torsten Frank and Frieder Christel
BMW Group

Sylvester Abanteriba
RMIT University

ABSTRACT

Modern exhaust systems contain not only a piping network to transport hot gas from the engine to the atmosphere, but also functional components such as the catalytic converter and turbocharger. The turbocharger is common place in the automotive industry due to their capability to increase the specific power output of reciprocating engines. As the exhaust system is a main heat source for the under body of the vehicle and the turbocharger is located within the engine bay, it is imperative that accurate surface temperatures are achieved. A study by K. Haehndel [1] implemented a 1D fluid stream as a replacement to solving 3D fluid dynamics of the internal exhaust flow. To incorporate the 3D effects of internal fluid flow, augmented Nusselt correlations were used to produce heat transfer coefficients. It was found that the developed correlations for the exhaust system did not adequately represent the heat transfer of the turbocharger. This paper addresses the fluid flow phenomena present in the turbine volute and applies augmented Nusselt correlations to accurately represent the heat transfer coefficients of the internal volute surface. Due to the broad range of operating conditions that are applicable to the turbocharger and the varied states of fluid flow that occur, algorithms are used to apply the appropriate Nusselt correlations and augmentations. Furthermore, the turbocharger extracts enthalpy from the working fluid; therefore to accurately calculate surface temperatures of downstream components and that of the turbocharger itself, an energy extraction model is used. Validation was conducted with four vehicle configurations. The hot-end of each configuration was aimed to be distinctly different to test the robustness of the prediction model. A tolerance range of +50/-20K was used for the study, however temperature differences were generally well within the tolerance range.

CITATION: De Vos, S., Haehndel, K., Frank, T., Christel, F. et al., "The Development of Turbine Volute Surface Temperature Models for 3D CFD Vehicle Thermal Management Simulations: Part 3: Exhaust Radial Turbine Volute Systems," *SAE Int. J. Passeng. Cars - Mech. Syst.* 7(2):2014, doi:10.4271/2014-01-0648.

INTRODUCTION

For modern automotive exhaust systems, turbochargers are common place. Due to their ability to downsize engines and reduce carbon emissions, the turbocharger is used in most BMW engine configurations. The thermal behavior of the turbocharger is important to the thermal management of the vehicle as it is located near thermally sensitive components.

In the pre development process, the thermal behavior of the vehicle is evaluated to ensure thermal safety and endurance of components. The turbocharger and the exhaust system that it is a constituent of, is a major contributor to the thermal behavior of the vehicle.

The effect of the turbocharger on downstream components is also of importance to the engineer. The enthalpy extracted from the exhaust gas affects the energy balance of conjugate heat transfer of exhaust components. The performance of

thermally dependant components, such as the catalytic converter is also dependant on the upstream turbocharger. At the early pre development stage, internals of the turbocharger are not always available. Therefore loss modeling and entropy generation are not adopted due to the requirement for detailed internal geometrical parameters. An alternative is to use a simple Brayton cycle analysis. Pressure ratios and isentropic efficiency factors are derived from experimentally developed performance maps of similar configuration engines.

A previous study [1] developed a methodology where the 3D phenomena of flow could be applied to a 1D fluid stream and therefore satisfy the boundary conditions for a thermal simulation. This study produced good correlation with experimental data and significantly decreased the computational demand of thermal simulations. The use of convection augmentation factors (CAF) in conjunction with Nusselt number correlations allowed accurate relations to be formed between physical phenomena of flow and heat transfer. Augmentations were applied for pulsation, entrance effects, surface friction and pipe bend effects. In the turbine volute section the flow phenomena is distinctively different to that of internal flow in other exhaust components. Therefore relations of flow characteristics to heat transfer must be developed to represent the unique and complex flow of turbocharger volute systems.

Due to the complexity of the flow in turbine volutes, it has been stated in literature, that it is difficult to predict heat transfer coefficients (HTC) in a simple manner [2]. Unsteady inlet flow conditions to the turbocharger are dependent on the engine operation state. The geometrical complexity of the upstream manifold causes chaotic rotation in the fluid. As a result, the relation between the fluid flow characteristic and HTC's via Nusselt correlations are difficult.

The volute section in a radial turbine is a complex shape. For a given aspect ratio of a turbine volute, the internal diameter and the radius of curvature attenuates proportionally. The basic geometry of a turbine volute is shown in [Figure 1](#).



Figure 1. Typical turbine volute geometry in CAD.

Flow through the volute is subjected to a centrifugal force due to the curvature in the flow path. With the assumption that the turbine volute flow does not interact with the turbine impeller or the guide vanes and steady inlet conditions, it is proposed that modeling flow can be achieved by implementing secondary Dean Flow on the fluid.

DEAN SECONDARY FLOW

Dean flow characterizes the effect of centrifugal flow on fluid through a curved pipe. In the early study by W. R. Dean, 1928 [3] the Dean number was derived from the perturbation of steady flow in a straight pipe with small radiuses of curvature. From W.R. Dean's work, the non-dimensional dean number was developed.

$$De = Re \cdot \left(\frac{r}{R}\right)^{\frac{1}{2}} \quad (1)$$

Where:

Re - Reynolds Number, $\frac{\rho u L}{\mu}$

r - Inlet Radius of flow path

R - Radius of Curvature

The secondary flow pattern causes an augmented velocity distribution from the inner convex wall to the outer concave wall. In laminar steady state flow the secondary flow is two counter rotating cells as shown in [Figure 2](#).

The Dean number application was extended to higher curvature to inlet radius ratios $\delta = \frac{r}{R}$ and with more realistic inlet conditions, including turbulent flow. The observation of the secondary flow superimposed on the primary flow is shown in multiple studies [4]. The earliest was conducted before the work of W. R. Dean [5]. A numerical study [6] showed through Fourier-series expansion, higher dean numbers incur stronger secondary flow patterns.

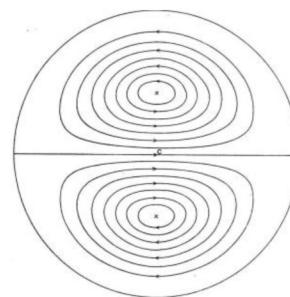


Figure 2. representation of two dean vortex shells [3]

For turbulent flows and high dean numbers, the formation of upto six cells have been observed dependent on the flow conditions [6]. The development of the secondary flow through curved pipes has been well covered in literature. Correlations for the development length is presented in the work of F.T smith, [7] who concluded that for turbulent flow with high dean numbers, the required length corresponding to $\sqrt{rRDe}$ [6]. It is noted that in literature, the dean number is defined differently. In the work of smith, the dean number was defined as

$2\sqrt{\frac{r}{R}\left(\frac{2RU_b}{\nu}\right)}$. The development length was observed to be smaller than the length required for boundary layer development in straight pipes. The centrifugal force imposed on the flow is known to induce stability in the flow field. Observation of this phenomenon is seen in a delayed critical Reynolds number for transition. Studies have been conducted to understand the relation between and the increased critical Reynolds number [8; 9]. The later study [9] presented a correlation for Re_{crit} . This is presented in equation 2.

$$Re_{crit} = 2100 \left(1 + 12 \left(\frac{d}{D} \right)^{\frac{1}{2}} \right) \quad (2)$$

Where d and D is internal and curvature diameter respectively.

Heat Transfer Modelling

The subject of heat transfer for Dean flow is popular for its uses in heat exchangers. Due to the centrifugal force, the boundary layer is affected, therefore enhancing heat transfer. A velocity distribution is shown in figure 3 to visualize the perturbation to the boundary layer.

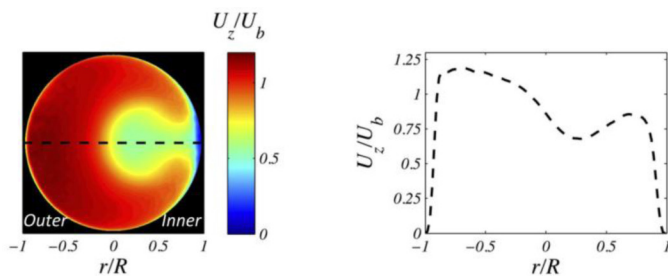


Figure 3. Example of velocity distribution over the cross section of a 90° pipe bend relative to bulk flow. [4]

From Figure 3 U_z is defined as the streamwise velocity and U_b is the bulk velocity. The profile of streamwise velocity scaled by bulk velocity along the horizontal axis is presented on the right.

Many studies have been performed for helically coiled pipes. In such systems, pitch is present in the geometrical configuration. This causes torsion to the flow. It was found for the characterization of low Prandtl numbers, torsion does not yield a significant effect on heat transfer [10]. Therefore for low Prandtl number correlations the application of helically coiled pipe Nusselt correlations is valid.

In order to encompass the entire operating range of the turbo charger multiple correlations were surveyed from literature. Correlations that satisfied the criteria for application to the turbine volute were chosen. Criteria were based on Prandtl number, Reynolds range and Dean range. Three Nusselt correlations were used to represent the relation between flow characteristic and heat transfer.

Laminar Flow Heat Transfer

In laminar flow regimes the secondary flow patterns poses a high augmentation to the heat transfer. For laminar cases, a study by Kalb and Seader, were able to solve the Navier-Stokes equations to determine the effect of fully developed Dean flow on heat transfer [11]. The correlation developed in the study is applicable to Prandtl numbers as low as 0.7. Their study was based on a curved pipe with uniform-wall temperature. The correlation was presented from the study shown in Equation 3.

$$Nu = 0.836De^{0.5}Pr^{0.1} \quad (3)$$

For

$$De > 80$$

Application of this correlation is for flow laminar flow regimes. The transition Reynolds number is specified from equation 2.

Turbulent Flow Heat Transfer

For turbulent flow the energy of the fluid is considerably higher, as a result the boundary layer is thinner than that in laminar flow. The influence to the heat transfer coefficient is less prevalent compared to laminar flow [12]. However studies have shown enhanced Nusselt Numbers for turbulent curved pipe flow, boundary layer influences are still significant [12,13].

R. Seban & E. McLaughlin [13] cited the experimental results of M. Adler, 1934 [14] to base additional experimentation and extend the understanding of heat transfer coefficients for turbulent flow through curved pipes. A well-known correlation was developed from the study which is shown in equation 4. This study was performed for both small and large coils and encompassed a wide range of turbulent Reynolds numbers up to the magnitude of 50000.

$$Nu = 0.023Re^{0.8}Pr^{0.4} \left(Re^{\frac{1}{20}} \left(\frac{r}{R} \right)^{\frac{1}{10}} \right) \quad (4)$$

For

$$6000 < Re < 50000$$

Turbulent Flow-High Reynolds Heat Transfer

In high Reynolds number flows, the boundary layer is thin and heat transfer coefficients are enhanced. The effect of the Dean secondary flow alters the boundary layer to enhance the convection to the surface of the pipe. The literature presents a phenomenon where at excessively high Reynolds numbers; the augmentation of the secondary flow is no longer significant [15]. This is presented in Figure 4.

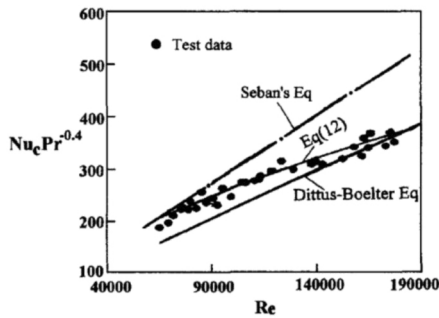


Figure 4. The attenuation of Nusselt numbers from curved pipe heat transfer to the Dittus boelter correlation for straight pipes at high Reynolds. (L. Guo et al, 1998)[15].

To model this flow, a commonly accepted Nusselt correlation developed by B.S. Pehtukov, 1970 which is used for straight pipes is utilized [16]. This relation is presented in Equation 5.

$$Nu = \frac{\left(Pr Re \left(\frac{f}{8}\right)\right)}{1.07 + 12.7 \sqrt{\frac{f}{8}} \left(Pr^{\frac{2}{3}} - 1\right)} \quad (5)$$

For

$$Re > 60000$$

The curvature of the pipe also influences the friction factor. Experimental investigations over a wide range of curvature ratios and Reynolds numbers were conducted by Ito, 1959 [8]. In this study, correlations for the Darcey-Weisbach friction factor, f was derived for both laminar and turbulent flow regimes.

$$f_{turb} = 0.304 Re^{-0.25} + 0.29 \sqrt{\delta} \quad (6)$$

CAF - Secondary Flow Development

Literature states that in the flow immediately downstream of the pipe bend, significant azimuthal secondary flow is induced in the boundary layer and at the pipe axis flow is transferred from the outer wall to the inner wall of the pipe bend due to the pressure gradient [17]. However a length is required for the secondary flow to fully develop [18]. Thus the application of Nusselt correlations that link fully developed Dean Flow and heat transfer would over predict the heat transfer in the application of the turbine volute.

The Dean number was developed to characterize the characteristic magnitude of the secondary flow. The Reynolds number has been used to characterize the flow regime in the primary flow direction. In this study an augmentation factor is proposed to reduce the Nusselt number to represent the partially developed secondary flow effects on the heat transfer coefficient. This augmentation factor is proposed as the ratio of

the Dean number to the Reynolds number, ultimately being a non-dimensional ratio of the inlet radius and radius of curvature as shown in equation 7.

$$CAF_{ent} = \frac{De}{Re} = \frac{\left(\left(\frac{\rho u L}{\mu}\right) \sqrt{\frac{r}{R}}\right)}{\frac{\rho u L}{\mu}} = \sqrt{\frac{r}{R}} \quad (7)$$

CAF - Pulsation

The flow proceeding from a reciprocating engine is inherently unsteady due to the cyclic excersion of gases and valve lifting. Pulsation has an effect on the heat transfer coefficients. Literature has presented conflicts in regards to fluid pulsation enhancing the heat transfer coefficient, without the complexity of superimposed secondary flow [15]. The assumption is made that the secondary flow is influenced by the pulsation and therefore augmentation factors used in straight pipe flow are applicable to the turbine volute. Complications in the effect of pulsation on heat transfer arise by multiple factors contributing to heat transfer augmentation. From literature, both frequency and velocity amplitude of the pulsation contribute to the ehancement of heat transfer [19]. The study by Wang and Zhang [19] produced relations between CAF, Reynolds number and Wormsley number. The study also showed the effect of velocity amplitude on CAF values are also presented.

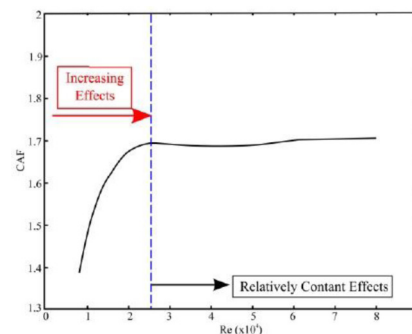


Figure 5. Pulsation Reynolds based CAF. (Wang and Zhang, 2005) [19].

Given the geometry of modern exhaust manifolds the pulsations of exhaust gas are superimposed to increase the frequency; this intern reduces the velocity amplitude. It is assumed that with the geometrical properties of the exhaust system and the operating range of engines, the velocity amplitude is limited to 1.5 [1]. As a result the phenomenon of recirculation is reduced and is applicable to the Nusselt correlations. From figure 6 it can be observed that the effects of velocity amplitude are relatively constant below the velocity Amplitude of 1.5. Thus the pulsation CAF variation over the operating range is based on the Reynolds number and Wormsley number.

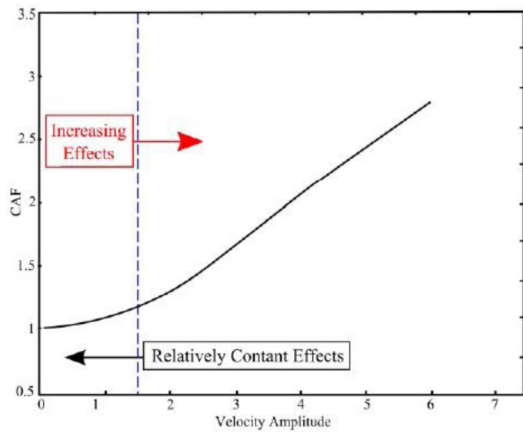


Figure 6. Pulsation CAF augmentation due to Velocity Amplitude (Wang and Zhang, 2005) [19]

Energy Augmentation

When dealing with pre development vehicles, often more intricate details of component geometry is not available. As a result, computational modeling must incorporate generalizations to represent the absent geometrical parameters. For the turbocharger, internal geometry is specified much later in the design phase of the vehicle. Therefore methods such as energy loss modeling proposed by Denton, 1999 [20] are not feasible for this application due to their requirement of internal geometrical parameters. Alternatively, a Brayton cycle analysis is employed. Pressure ratios and efficiency factors were derived from performance maps and applied to [Equations 8, 9, 10, 11, 12](#). Pressure ratios and efficiency factors were utilized from similar configuration vehicles. It is noted that a relatively high variation in gas temperature dose not significantly affect the thermal resistance network given the high temperature gradients of the component and gas temperature. Energy balance equation is given in [Equation 8](#).

$$\frac{dE_{cv}}{dt} = \dot{Q}_{cv} - \dot{W}_{cv} + \sum_i \dot{m}(h_i + \frac{V_i^2}{2} + gz_i) - \sum_e \dot{m}(h_e + \frac{V_e^2}{2} + gz_e) \quad (8)$$

Applied to a turbine:

$$\dot{m}h_{in} + \dot{Q}_{in} = \dot{m}h_{out} + \dot{W}_{out} \quad (9)$$

With the assumption of adiabatic operation:

$$\dot{W}_{out} = \dot{m}(h_{in} - h_{out}) = \dot{m}Cp(T_3 - T_4) \quad (10)$$

Isentropic expansion without entropy generation can be described via [equation 11](#).

$$\frac{T_4}{T_3} = \left(\frac{P_3}{P_4}\right)^{\frac{k}{k-1}} \quad (11)$$

The efficiency factor of the turbine can be defined by:

$$\eta_t = 1 - \frac{q_{out}}{q_{in}} = 1 - \frac{Cp(T_3 - T_2)}{Cp(T_4 - T_1)} \quad (12)$$

Thus [equation 11](#) becomes:

$$\frac{T_4}{T_3} = \left(\frac{P_3}{P_4}\right)^{\eta_t \left(\frac{k}{k-1}\right)} \quad (13)$$

METHODOLOGY

This section of the document specifies how the investigation was conducted. It comprises of two sections, computational methods and experimental methods.

Computational Methods

The implementation of predicted internal heat transfer coefficients is used in an alternative method to conventional “all-in-one” thermal simulations. Coupling of external convection to a thermal model with virtual thicknesses allows for a much lighter calculation and therefore reduced computational demand. In this investigation, Radtherm is used as the thermal solver and Star-CCM+ is used resolve the fluid dynamics.

In this thermal simulation process, external convection is mapped onto the thermal model after calculation in Star-CCM+. The internal flow is modeled using a 1D fluid stream. Heat transfer coefficients for internal surface of the exhaust are predicted from the models developed in the work of K. Haehndel, 2009. The heat transfer coefficients developed for the turbine volute are derived from the equations specified earlier in this paper. Physical modeling of components is conducted in Radtherm specifying thicknesses, material properties and surface conditions.

In order to keep the calculation of external convection accurate, the surface temperatures from the thermal simulation are mapped onto the CFD simulation model in a global cycle to achieve a converged heat transfer coefficient. This process is presented in [Figure 7](#).

Entire Under-Body Heat Transfer Process



Figure 7. Coupling Process for thermal simulation between Radtherm and Star-CCM+.