Proceedings of IMECE2002
ASME International Mechanical Engineering Congress & Exposition
November 17–22, 2002, New Orleans, Louisiana

IMECE2002-32095

CFD ANALYSIS OF LIQUID-COOLED EXHAUST MANIFOLDS IN A REAL ENGINE CYCLE

Uros Kresovic, Wallid Hussein, and Chenn Q. Zhou
Purdue University Calumet, Department of Engineering
Hammond, IN 46323

Jim Majdak and Robert Cantwell
Hadady Corporation, 510 W 172nd Street
South Holland, IL 60473

ABSTRACT

Liquid-cooled exhaust manifolds are used in turbocharged diesel and gas engines in the marine and various industrial applications in order to minimize heat rejection to surrounding areas, maximize energy to the turbocharger, and maintain a maximum allowable skin temperature. A commercial CFD software FLUENT® was used to analyze liquid-cooled exhaust manifolds in a real time engine cycle. Detailed information of flow property distributions and heat transfer were obtained in order to provide a fundamental understanding of the manifold operation. Experimental data was compared with the CFD results to validate the numerical simulation. Computations were performed to investigate the parametric effects of operating conditions on the performance of the manifold. Two different geometries were compared. One of them was found to have better performance, resulting in an approximately 2 to 3% fuel consumption improvement.

INTRODUCTION

Liquid-cooled exhaust manifolds are used on turbocharged diesel engines for marine, oilrig, and other industrial applications. The purpose of cooling the manifold is to prevent possible fire hazards by maintaining a maximum safe temperature on the exposed surfaces or "skin" of the manifold. The adverse effect inherent to cooling the manifold is a loss in engine efficiency. Modifying manifold design is one way to improve efficiency. Hadady Corporation has been manufacturing liquid-cooled exhaust manifolds for many years. The designs, however, have been based on intuition, prior experience, and trial and error due to the complex geometry and physical phenomena. With the rapid advance in computer technology and numerical methods, CFD has become a powerful tool used to provide information on parametric effects for optimization of manifold design and performance. Hadady Corporation has identified CFD as a useful and cost-effective tool for the analysis and design of liquid cooled exhaust manifolds.

A similar diesel application would be the manifold on a turbocharged truck engine. The purpose of the exhaust manifold here is also to route the exhaust gases to a turbocharger, but liquid cooling is not required. Turbochargers use the exhaust gas to drive a turbine. The work produced by the turbine is then used to compress intake air resulting in higher overall engine performance.

Past studies have only dealt with standard exhaust manifolds due to the prevalence of automotive engines. The focus of those studies was mainly in the manifold junctions. For example, Sierens and Snauwaert used Laser-Doppler anemometry and a numerical simulation model based on the method of characteristics to study the flow patterns in compact manifold junctions [1]. Two simple junctions were studied in order to determine empirical coefficients for their numerical method. Around the same time, Morimune and Hirayama recognized the importance of the flow patterns in exhaust manifolds for matching turbo chargers to diesel engines [2]. Several numerical models were developed and revised to predict pressure, temperature, and mass flow at T-junctions in the exhaust system. More recently, Payri et al studied the influence of Y-junction geometry in the exhaust system using an experimental characterization method and a one-dimensional gas dynamic model [3]. The flow through junctions in all exhaust manifolds is of great importance; however these studies only looked at simplified, single junctions in standard exhaust manifolds. Computational resources and time were the main reasons for the simplifications. Liquid-cooled exhaust manifolds are very complex and have multiple junctions as well as many other flow obstructions.

Another method commonly used analyzes standard exhaust systems by studying heat transfer. The heat transfer from the exhaust system has been extensively studied in recent years [4,5]. Zhang et al used a finite volume method to study the heat transfer in the exhaust system [6]. Several different walls were used in the study including a single wall pipe and an air gap pipe. The air gap pipe was shown to reduce the outer pipe skin temperature while maintaining high gas temperature. This is a possible alternative to liquid cooling that could greatly improve engine efficiency, however, the skin temperature would be more difficult to regulate under that design. Again, liquid-cooled exhaust manifolds were not studied and simplified geometries were used.

In this study, the commercial CFD software, (FLUENT®) was used to analyze liquid cooled exhaust manifolds in a real time engine cycle [7]. The objectives are to provide detailed information of flow property distributions and heat transfer for fundamental understanding of the manifold operation, and to perform parametric studies for improving the manifold performance. The three-dimensional unsteady state model of the exhaust manifold was examined. Experimental data was compared with the CFD results to validate the numerical simulation. A number of computations were performed to investigate the parametric effects of operating conditions on the performance of the manifold.

THEORY AND APPROACH

The manifold in this study was designed for use on a marine engine. The exhaust manifold is a component that reaches very high temperatures. The temperature of the combustion products that go through the exhaust manifold is, in this example, 964 K. Without control over the manifold surface temperature, the heating of the air in the boat's engine compartment can be severely dangerous. High temperatures in the boat's compartment can cause fires, deterioration of seals, gaskets, and wiring, and engine overheating. One way of reducing surface temperature is through the application of a concentric tube heat exchanger. In this type of heat exchanger, a fluid is passed through the outer shell of the exhaust manifold to induce heat transfer from the inner tube. Hadady Corporation has used this method to reduce the surface temperature of the exhaust manifold.

To analyze the flow and heat transfer patterns in a manifold, a commercial software, FLUENT®, was used. It is a state-of-the-art for general purpose CFD software and has been widely used in industry to solve fluid flow problems. In this research, the focus is on the exhaust part of the manifold and a three-step approach was followed in order to meet the objectives.

The first step was to establish a baseline case in FLUENT®. This included identifying the geometrical and operational conditions of an existing exhaust manifold from Hadady Corporation, creating meshes, setting up boundary conditions, and performing simulations.

The second step included analyzing and validating the results. This provides detailed flow property distributions that are not physically obtainable by testing but are very useful for fundamental understanding of the manifold operation. To ensure realistic and accurate results, validations were made by comparing computational results with experimental results obtained from Hadady Corporation

The third step was a parametric study to improve the manifold design and to optimize the performance. Effects of surface roughness, coolant temperature, and geometry on the flow and heat transfer characteristics and enthalpy availability were investigated. During the parametric study, only one parameter of the fluid flow was changed while all others were held constant.

NUMERICAL METHODS

Geometry and Grids

The geometry was obtained from Hadady Corporation and constructed using Autodesk® Mechanical Desktop® v5.0 for a full three-dimensional representation of the actual geometry. Figure 1 shows the exhaust manifold volume that served as the baseline case in this research. The most important characteristics of the geometry can be noted on the figure. The exhaust manifold is used on an in-line six-cylinder engine, and therefore, has six inlet ports and one outlet. The exhaust manifold's outlet is connected to the turbocharger that is used, as previously mentioned, to drive the compressor, which compresses incoming air.

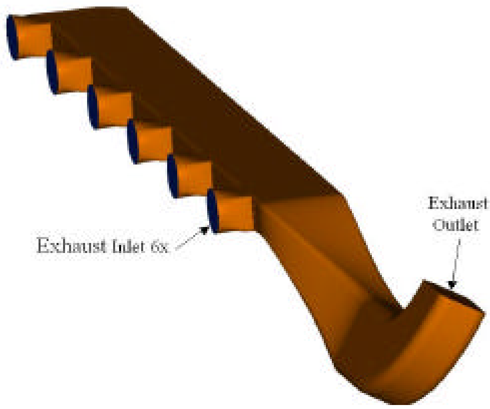


Figure 1: Model of the Exhaust Manifold Volume

The computational grids were created using Gambit®, a preprocessor to create geometrical models and meshes used for numerical solutions in the FLUENT® program. In this research, the tetrahedral mesh was used because the geometry consisted of many small three dimensionally curved faces, especially on the inlets. Grid sensitivity studies were conducted to select the number of the nodes and the results will be presented in a subsequent section.

Boundary Conditions

Flow conditions are needed to specify boundary conditions. Table 1 shows the data obtained from Hadady Corporation and was used to define boundary conditions.

Table 1: Typical Flow Conditions

Engine Speed	2300 rpm
Outlet Pressure	312000 Pa
Inlet Temperature	964 K
Exhaust Wall Temperature	589 K
Exhaust Duration	275°

Because this study focused on the exhaust part of the manifold, the exhaust wall temperature was specified to simulate the effect of liquid coolant. It was determined based on the log mean temperature in consideration of the heat transfer between the exhaust gas and the coolant liquid.

In order to consider unsteady state conditions, transient inlet velocities have to be specified. The function of the inlet velocity to time was obtained from Hadady Corporation to represent the movement of the exhaust valves. When the exhaust valve opens, the exhaust velocity approaches 150 m/s; when the valve is closed, the inlet velocity is zero. This velocity profile imported into FLUENT® and used as the inlet velocity boundary condition.

Time Steps

The primary goal of this research was to obtain the unsteady state analysis of the three-dimensional model of the exhaust manifold. The results obtained from Hadady Corporation and presented in Table 1 were measured for an engine test speed of 2300 rpm (revolutions per minute). Based on an arbitrary 5° crank rotation increment, a time step was calculated and is defined throughout this paper to be 0.000362 seconds. The distributions of flow properties at every time step were computed and saved for analysis.

RESULTS AND DISCUSSION

Grid Sensitivity Study

Grid sensitivity studies were performed to determine a proper grid size to apply for all simulations. Several grids were created and one of them was selected to give a grid independent result in combination with the shortest computational time.

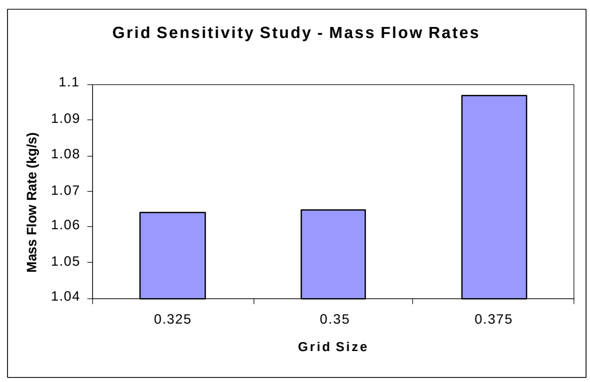


Figure 2: Grid Sensitivity Study – Mass Flow Rates

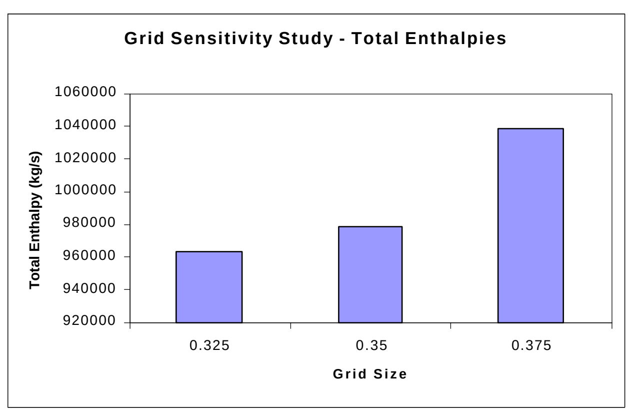


Figure 3: Grid Sensitivity Study – Total Enthalpies

The most important parameters of fluid flow in this study are mass flow rate and total enthalpy. Their changes were compared for mesh size of 0.325, 0.35, and 0.375 cubic inches. The differences in mass flow rates are presented in Figure 2. The differences in total enthalpies are presented in Figure 3.

From these figures one can see that a mesh size of 0.35 cubic inches can be used for all parametric studies and calculations.

Validation

Even if converged computational results can be obtained, CFD analysis may not be correct without validations. This can happen when the boundary conditions are not set up properly, when the grid distribution is not proper, or when an inappropriate assumption is made. Therefore the data obtained from the CFD analysis must be compared with the experimental data. In this study, the mass flow rate at the outlet of the exhaust manifold measured by Hadady Corporation was used for validation. The comparison between the computed mass flow rates of the baseline case and measured data is given in Table 2. It can be seen that the difference is approximately 1.3 %, which is acceptable. The flow rate was held constant throughout the analyses. More verifications and validations can be found in a previous study [8].

Table 2: Comparisons between Calculated and Measured Data

Calculated Mass Flow Rate (kg/s)	Measured Mass Flow Rate (kg/s)	Percentage Difference (%)
1.064	1.05	1.3

Flow Characteristics

Unsteady state analysis implies changes of flow patterns in time. Computational results of flow characteristics at different time were recorded and displayed as a movie to provide visualized pictures of flow patterns for fundamental understandings. In the paper, it is very hard to represent results that change in time rapidly, in every moment. In order to represent some patterns of fluid flow, results for two time steps are presented. These two time steps are selected to represent two important moments. The first is at the 250th time step when three cylinders are open, and the second is at the 335th time step when two cylinders are open.

Figure 4 displays velocity, temperature, and pressure contours at a 2-D cross section for the 250th time step. The scale is indicated by a color bar. The red color represents the highest value and the blue represents the lowest value. At this moment three inlets are opened: the third, fifth and sixth. Cylinder number 1, or the front of the engine, is at the bottom. The turbocharger is mounted at the rear. Clearly, velocity, temperature, and temperature distributions demonstrate corresponding patterns. Note the high velocity area near cylinders 5 and 6, where the mass flows of three different cylinders converge.

Figure 5 displays velocity, temperature, and pressure contours at a 2-D cross section for the 335th time step. At this moment two inlets are opened: the first and fourth. Again, velocity, temperature, and temperature distributions reflect corresponding patterns.

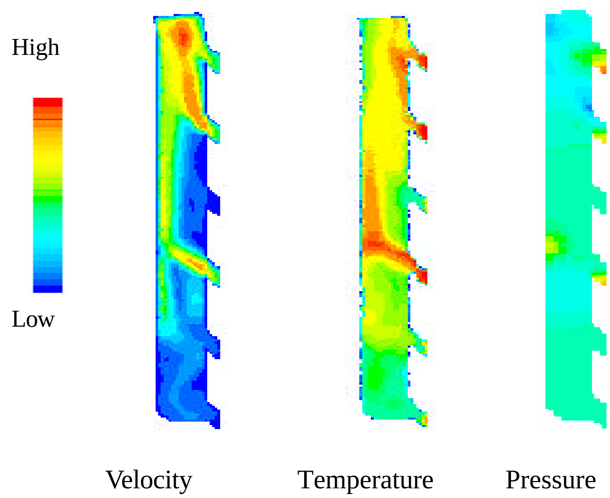


Figure 4: Flow Property Distributions through the Center of Exhaust Log at Time Step when the Exhaust Valves of Three Cylinders are Open

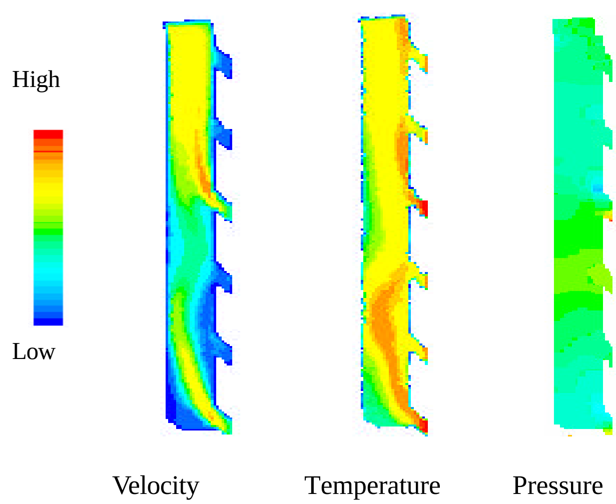


Figure 5: Flow Property Distributions through Center of Exhaust Log at Time Step when the Exhaust Valves of Two Cylinders are Open

Parametric Studies

Once a validated model has been achieved, parametric studies can be performed. During the parametric study, one parameter of the fluid flow was changed while the others are held constant. This is a real advantage of the CFD analysis. The parameters varied in this study include geometry, surface temperature, and surface roughness. Two geometries with different designs were compared. Three different surface roughness values were examined and five different applied wall temperatures were investigated.

Total enthalpy at the outlet of the manifold is used to evaluate parametric effects. It provides a measure of the ability of the exhaust gas at the outlet of the exhaust manifold to produce work. If the total enthalpy at the outlet is higher, the turbocharger's performance will be better. Consequently, the fuel efficiency is higher and the engine's performance is better.

Effects of Geometry

Two geometries were compared in this research. Both are existing manifolds designed by Hadady Corporation. The baseline case (prototype 1) is the old design and the other case is the modified design (prototype 2), which has smoother transitions in the areas of inlets and the outlet in geometry comparing to the baseline case as shown in Figure 6.

Prototype 1 Prototype 2

Figure 6: Different Geometries of Manifolds

Figure 7 presents the results of total exit enthalpies at different times for the two geometries. The average total enthalpies at the outlet over a real engine cycle are given in Table 3. Figures 8 and 9 show the velocity and temperature distributions for both prototype 1 and prototype 2 at a at time step when the exhaust valves of two cylinders are open. The figures show Prototype 2 retains more heat inside the exhaust log, and has a more uniform velocity distribution. Clearly, the new geometry with smoother transitions showed better performance in terms of increased enthalpy than the baseline case. The total enthalpy at the outlet of the new geometry is 3.76% higher than that of the baseline geometry as shown in Table 3. This increase of the enthalpy is because the smoother transitions induce more uniform flow pattern and consequently less energy is consumed. This result is very encouraging and consistent with the experimental observation. In the engine test, it was found that the prototype 2 had 3% fuel consumption savings compared to the prototype 1.

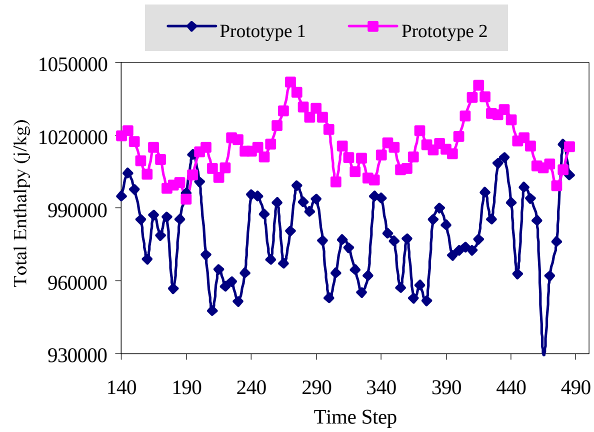
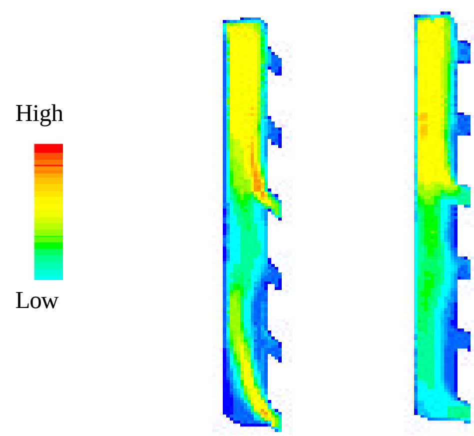


Figure 7: Total Exit Enthalpies for Different Geometries

Table 3: Total Enthalpies for Different Geometries

Prototype	Total Enthalpy (j/kg)	%Difference
1 (Baseline Case)	978759	0.0
2 (Modified Case)	1015597	+3.76



Prototype 1 Prototype 2

Figure 8: Velocity Distributions for Two Different Geometries

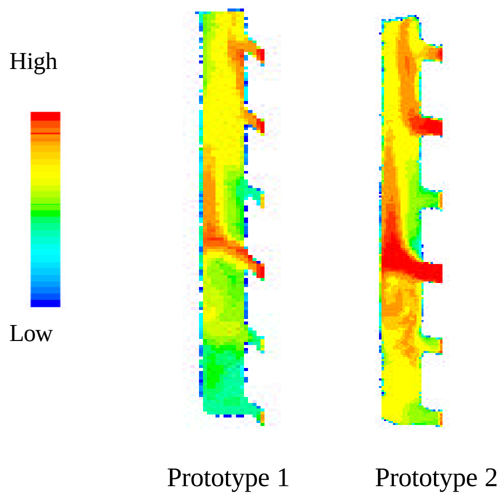


Figure 9: Temperature Profile for two Geometries

Effects of Surface Roughness

Surface roughness is a parameter that has a strong influence on the flow characteristics of pipes. The issue when discussing surface roughness is to find relationship between surface roughness and the manifold's performance. That information is valuable in determining the cost/benefit relationship between alternative designs.

Three different surface roughness values from 0.0 inches to 0.0102 inches were investigated. The results are presented in Figures 10 and 11.

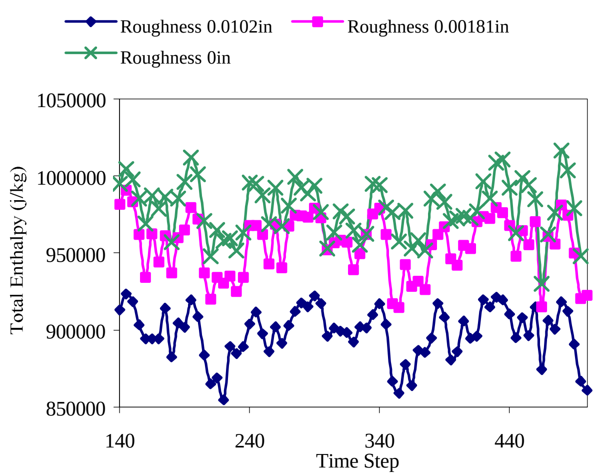


Figure 10: Total Enthalpies for Different Surface Roughness

Figure 10 shows the results of total exit enthalpies at different times for different surface roughness. Figure 15 shows the relationship between total enthalpy at the outlet and surface roughness. It can be seen that the total enthalpy is larger for lower surface roughness values, which was expected. The smoother the surface, the lower the energy loss. Table 7 shows the difference between the smallest and the largest surface roughness is approximately 8%. Figure 12 shows the temperature distributions for the smooth surface and a rough surface with the surface roughness of 0.0102 inch. The smoother surface gives more uniform temperature distribution and has less energy loss.

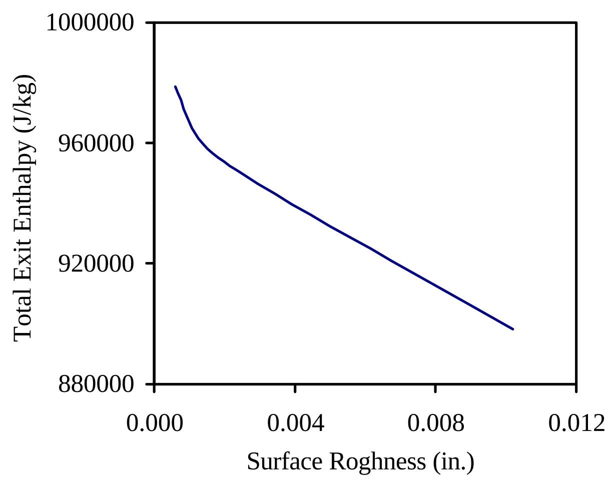


Figure 11: Effects of Surface Roughness on the Total Enthalpies

Table 4: Comparisons of Total Enthalpies at Different Surface Roughness Values

Surface Roughness (in)	Total Enthalpy (j/kg)	%Difference
0.0 (Baseline Case)	978759	0.0
0.00181	955136	-2.42
0.0102	898049	-8.25

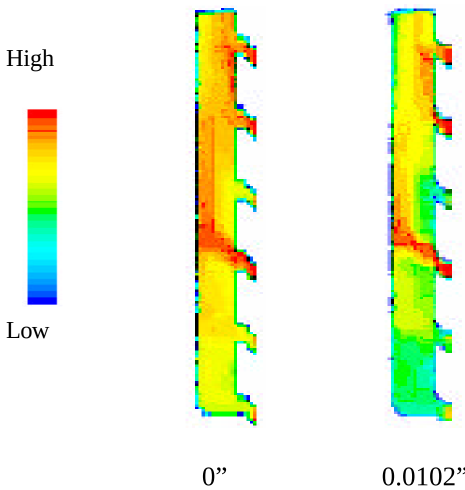


Figure 12: Temperature Distributions for Two Different Surface Roughness Values

Effects of Wall Temperature

Controlled wall temperature can have an impact on an exhaust manifold’s influence on engine performance. Liquid-cooled exhaust manifolds are used to route products of combustion from engine cylinders to turbochargers at a limited skin temperature. The exhaust manifold is a component that reaches very high temperatures, because products of combustion that go through the exhaust manifold are at nearly 1000K for diesel combustion. The high temperatures cause the heating of air in the boat’s compartment that, in turn, can cause a severe danger to boat safety. In the liquid-cooled exhaust manifold designed by Hadady Corporation, the surface temperature is controlled by a fluid passing through the outer shell of the exhaust manifold to induce heat transfer from the inner tube.

In the calculations, a constant wall temperature was assumed to simulate the influence of a coolant that flows through the outer shell of the exhaust manifold. This assumption is reasonable because difference of the inlet and outlet coolant temperature is normally 20 K. In the baseline case, the value of the wall temperature is specified based on a log-mean temperature calculated from measured coolant temperatures at a typical condition. The coolant temperature and mass flow rate were determined by direct measurements at the engine manufacturer. What is not known is the relationship between coolant’s mass flow rate and temperature and total enthalpy at the outlet of the exhaust manifold. The hand calculations that would give a solution to this problem cannot be made without large assumptions that would lead to error because of the complexity of the geometry. FLUENT® provides a tool for estimating this relationship. By specifying different wall temperatures, the effects of coolant conditions can be found indirectly. The results of wall temperature parametric studies are shown in Figure 13 and summarized in Table 5.

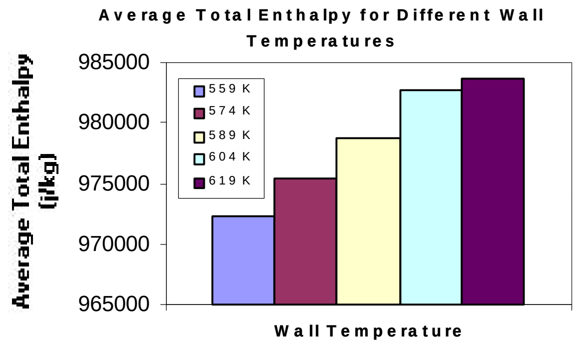


Figure 13: Total Enthalpies for Different Wall Temperatures

Table 5: Percentage Differences of Total Enthalpies for Different Wall Temperatures

Wall Temperature (K)	Total Enthalpy (j/kg)	Percentage Difference (%)
559	972297	-0.7
574	975477	-0.4
589 (Baseline Case)	978759	0.0
604	982714	+0.4
619	983738	+0.6

It is expected that when the wall temperature increases, the total enthalpy at the outlet increase because of less heat transfer.

CONCLUSIONS

Liquid-cooled exhaust manifolds in a real engine cycle were analyzed using a CFD software FLUENT®. The results provided visualized pictures of flow patterns inside the manifolds and parametric effects on the manifold performance.

It was found that increasing the surface roughness resulted in a decrease in the total enthalpy at the manifold outlet due to the increase of energy loss. The use of smooth surfaces, for example, material that is cold drawn rather than sand cast, would increase the enthalpy available to the turbocharger. On the other hand, increasing the coolant temperature resulted in an increase of the total enthalpy at the manifold outlet, because less heat was transferred between the coolant and the exhaust gas. Geometry also showed significant effect. A modified design with smoother transition provided higher enthalpy at the manifold outlet than the original, which corresponds to approximately a 2 to 3% fuel consumption savings on engine operations. These results are consistent with experimental observations. These conclusions are valuable to Hadady Corporation and its customers in gaining insight on the flow and heat transfer characteristics for optimizing the design and performance of their diesel engines.

REFERENCES

1. Sierens, R., and Snauwaert, P., 1987, "Study of the Flow Pattern in Compact Manifold Type Junctions by LDA.," ASME Transactions, Journal of Engineering for Gas Turbines and Power, Vol. 109:452-458.
2. Morimune, T., and Hirayama, N., 1988, "The Matching of Diesel Engine to Exhaust Turbocharger—Improved Calculation at T-Junction," ASME Transactions, Journal of Engineering for Gas Turbines and Power, Vol. 110:547-551.
3. Payri, F., Reyes, E., and Galindo, J., 2001, "Analysis and Modeling of the Fluid-Dynamic Effects in Branched Exhaust Junctions of ICE," ASME Transactions, Journal of Engineering for Gas Turbines and Power, Vol. 123:197-203.
4. Konstantinidis, P. A., Koltsakis, G. C., and Stamatelos, A. M., 1997, "Transient Heat Transfer Modeling in Automotive Exhaust Systems," Proceedings of the Institution of Mechanical Engineers, Journal of Mechanical Engineering Science, Vol. 211 Part C:1-15.
5. Kandylas, I. P., and Stamatelos, A. M., 1999, "Engine Exhaust System Design Based on Heat Transfer Computation," Energy Conservation & Management 40:1057-1072.
6. Zhang, Y., Phaneuf, K., Hanson, R., and Showalter, N., 1992, "Computer Modeling on Exhaust System Heat Transfer," Tenneco Automotive and the Society of Automotive Engineers, Paper No. 920262
7. FLUENT® User's Guide, version 6.0, (2002) Fluent Incorporated, New Hampshire.
8. Scheeringa, K., D. Schwerin, B. Groves, C. Q. Zhou, J. Majdak and R. Cantwell, "CFD Analysis of a Liquid-Cooled Exhaust Manifold", proceeding of 8th AIAA/ASME Joint Thermophysics and Heat Transfer Conference, St. Louis, Missouri 24 - 26 Jun 2002.