Experimental and Numerical Assessment of a Multi-Cylinder Engine Exhaust Manifold

Wisam Nasser Assi¹, Mohammed A. N. Ali² and Aseim S. Allawee³

1,2,3. Middle Technical University/Technical Engineering College-Baghdad

Abstract. This research presents an experimental and numerical analysis to examine the thermal loading of the exhaust manifold of a multi-cylinder gasoline engine operating under steady-state conditions. The local skin temperatures and surface heat fluxes variation are focused throughout the external surface of the exhaust manifold. A 3D modeling and simulation are employed in the numerical analysis using Solidworks software for modeling and ANSYS software for simulation. The generated numerical results are in good agreement with the measured exit gas temperatures and skin temperatures experimentally.

Keywords. Exhaust manifold, experimental testing, numerical modeling and simulation, skin temperatures.

1. Introduction:

The measuring or predicting heat transfer in the exhaust system in general and, especially, for the exhaust manifold in multi-cylinder engines for several reasons is extremely complex. Some of them are related to the nature of the flow in the exhaust manifold, which is characterized by high turbulence, even at low engine speed, the entrance region and its effect on heat transfer and flow. Also the high temperature of the exhaust-gas stream with strong fluctuations in temperature. Three heat transfer modes occur in the exhaust manifold; force convection between exhaust gas and inner manifold wall, heat conduction across manifold wall and free/force convection with ambient. Due to the great role played with exhaust manifold in improving engine performance and reducing emissions, many research studies have examined this important part in many aspects. Some of the researches are experimentally only, others theoretically and a third are combined the experimental and theoretical works. C. D. Bannister et al. (2011) [1], developed design of an exhaust system depended on the experimental description of heat transfer modes in a group of different pipe segments. Convective heat transfer relations for nine stainless steel exhaust bend segments with various wall thicknesses and radius were experimentally characterized over a scope of steady-state conditions. The result demonstrates that the exhaust gas temperature expectation for single-sheet curved sections displayed errors of less than ±5 percent over the main experimental test. G. C. Mavropoulos et al. (2009) [2], examined immediately the heat transfer process happening in both the exhaust manifold and cylinder...
head wall surfaces for an air-cooled diesel engine. Studied the significant influence of a transient event of speed and load variation in engine cyclic temperatures both for engine cylinder and exhaust manifold. The experimental investigation was carried out on the one-cylinder diesel engine. Compared the amplitude of skin temperature in cases steady-state and unsteady. The result illustration that skin temperature value was increased to 31°C higher than the corresponding ones which occurred through steady-state operation. B. Celikten et al. (2018) [3], The effect of improving the external heat transfer coefficient on the metal surface temperature of the exhaust manifold used in the heavy-duty engine was studied. The Co-Simulations method utilized to simulate ten 3D CFD gas flow analyses with a special characteristic. The experiments were performed on dyno engine cells with steady-state, full load measurements from (900-1900) revolutions per minute with 100 run increments that were taken in the dynamometer. The results show that the control point temperature prediction error was improved from 8% to 2.5% on an average and accuracy reached to 95.75% on average between Computer-Aided Engineering and experimental testing. K. Haehndel et al. (2013) [4], used correctional factors derived from a study of the specific influences on conjugate heat transfer modes within internal exhaust systems and exhaust gas dynamics. The entrance effects, engine induced pulsation, surface condition, and geometry are primary phenomena were found to have a significant influence on the internal heat transfer coefficients. S. Eroglu et al. (2016) [5], Computed the flow field inside the exhaust system and get temperature distribution at the special zone of the exhaust manifold area using computational fluid dynamics (CFD) analysis. The thermal analysis results represented by the distribution of temperature on the surface of the exhaust manifold were used to perform the structure durability analysis. The results demonstrate that Thermo-mechanical structural analysis is performed that predicts crack imitations through thermal fatigue test very well where the crack location was at center exhaust manifold area.

The present study was attempted to remedy this, by performing a comprehensive experimental study, which includes gas-temperature measurements at the inlet and exit sections with many external-surface-temperature measurements at nine locations in the tested exhaust manifold. This experimental work was assisted numerically by a comprehensive computational investigation performed using a commercial simulation software package ANSYS (Fluent) version R 19, to compare the experimentally measured distribution of skin temperature with the numerical generated results.

2. Plan of the work:

In this research, the two studies were carried out in the following order; firstly, the experimental testing was initially run the dynamometer engine in steady-state conditions with various speed and load operating conditions. Secondly, the numerical 3D Computational Fluid Dynamics (CFD) analysis was made based on the same necessary experimental boundary conditions that were provided, especially, in the worst-case thermal condition (engine speed and overloading). The experimental tests and the simulation are taking into account that the exterior of the exhaust manifold is subject to air forced convection applied by the engine fan. The experimental testing and numerical simulation results for the exhaust manifold skin temperature distribution are compared.

3. Experimental Analysis:

3.1 Engine and exhaust manifold

A spark ignition (SI) four-cylinder gasoline fuel engine type Mercedes-Benz 1993 equipped in the testing rig is used in the present work. The photo and schematic diagram of the engine testing rig are shown in Fig.1a, b. The specifications of the testing engine are listed in Table 1.
Figure 1. Experimental test rig: (a) Test rig photo, (b) The schematic diagram.

Table 1. Engine Specifications

| Engine Type       | Naturally Aspirated Petrol |
|-------------------|----------------------------|
| Engine            | Mercedes-Benz              |
| Cylinders         | Straight(Four-Stroke)      |
| Displacement      | 1997cm³                    |
| Bore x Stroke     | 89 x 80.25(mm)             |
| Compression Ratio | 9:1                        |
| Max. Power @ rpm  | 80kw (107.5hp)@5500 rpm    |
| Max. Torque @ rpm | 165 N·m (118 lb·) @ 3000 rpm |
| Fuel System       | Caruretor                  |
| Cooling           | Water                      |

Figure 2. Thermocouple locations on the exhaust manifold.
Exhaust gas-stream temperatures were estimated utilizing Type-K thermocouples with 2-mm distance across wire uncovered intersection. These thermocouples were roughly situated at the gas stream centerline at the passageway to every one of the short exhaust manifold runners and at the manifold outlet. A sum of 6 stream temperatures was estimated over the exhaust manifold. External manifold system skin temperatures were estimated utilizing 1.5-mm width Type-K thermocouple wire with high-temperature protection. The wires were spot-welded to the part surface to shape the thermal intersection. Each skin thermocouple had a development alleviation and was tied set up utilizing treated steel strips spot-welded over the protected wire. Outside skin temperatures were estimated in 9 zones over the manifold. Notwithstanding part temperature estimations, type-K uncovered wire thermocouples were utilized to quantify the fan stream air temperature radial neighboring the pivotal focal point of the manifold as shown in Fig. 2a, b.

3.2 Defining the boundary Conditions:

When the governing differential equations are set up to describe the problem in the domain, it is necessary to adjust the boundary conditions that specify the values of the mass flow rate and temperature similar to that considered in the experimental solution of the problem. The goal of the steady-state testing matrix was to capture a wide range of operation of the exhaust flow and temperatures under different engine speeds, loads and fuel flow rates. Table 2 describes the matrix of steady-state testing conditions.

3.3 Validation of Experimental Testing Rig:

In order to verify the data obtained from the experimental test facility for heat transfer. The tests were performed on a standard plain exhaust manifold. The experimental data compared with the results obtained from the well-known correlations (Dittus-Boelt, Colburn, Seider-Tate and Gnielinski’s equations) under the similar conditions to evaluate the validity of the exhaust manifold measurements shown in Fig.3, depending on the equations explained below. Quantitatively, the average absolute deviation between the measured and calculated values are 5.7% based on the Colburn equation and 21% relative to the Seider-Tate equation.

\[
Nu = 0.023Re^{0.8}Pr^{0.4} \quad \text{(Dittus-Boelte)}
\]

\[
Nu = 0.023Re^{0.8}Pr^{0.3} \quad \text{(Colburn)}
\]

\[
Nu = 0.027Re^{0.8}Pr^{1/3} \left( \frac{H}{\mu T} \right)^{0.14} \quad \text{(Seider-Tate)}
\]

\[
Nu = 0.016Re^{0.875}Pr^{1/3} \quad \text{(Gnielinski)}
\]
4. Numerical Analysis:

The computational fluid dynamic (CFD) is a numerical method that can be solved for real fluid flow, thus it has been widely used in a majority automotive manufacturers during the stages of design, research, and development of the internal combustion engine. Analytical solutions of the Navier-Stokes equations occur only for the simplest streams under ideal conditions. To achieve solutions for the real flow, the numerical method must be adopted whereby the equations are substituted by discretization equations that can be solved using numerical software [7]. In the presented research, a CFD finite volume method of analysis of the exhaust manifold was performed within ANSYS (Fluent) Workbench V R19 scheme. Before starting the simulation and due to the shape complexity, Solid work software was used to generate the physical model of the exhaust manifold in 3D. The dimensions of the exhaust manifold of the engine are recorded as reverse engineering. Then, with the good facilities of SOLIDWORKS software, the 3D model was created. The exhaust manifold of the engine consists of four 37 mm inner diameter, among which the first cylinder exhaust ports and the fourth were linked together, and the second port was linked with the third, then the couples above are connected together. This form is termed as a 4-2-1 exhaust manifold, as shown in Fig. 4a, b. The boundary conditions for the numerical simulation were described in detail in this research as preprocessing (model generating, meshing, solid and fluid materials properties, type of flow and thermal loadings). The most widely used method to check the solution convergence is the error residuals, which is directly quantified the error in the solution of the system of equations. The solution converged when the residuals are below a set tolerance limit. This limit was set to be about $10^{-4}$ for all variables except the energy equation residual must be less than $10^{-6}$.

Table 2. Testing operation

| Engine speed (RPM) | Load (N.m) | Fuel (g/sec) |
|--------------------|------------|--------------|
| 1800               | 12         | 0.635        |
|                    | 22         | 0.918        |
|                    | 34         | 1.073        |
| 2200               | 20         | 1.044        |
|                    | 32         | 0.99         |
|                    | 41         | 1.622        |
| 2600               | 37         | 1.772        |
|                    | 51         | 2.381        |
|                    | 62         | 2.823        |
| 3000               | 39         | 2.006        |
|                    | 54         | 2.541        |
|                    | 67         | 3.176        |

Figure 3. Nusselt number vs. Reynolds number for measured data and calculated data from standard correlations.
The complex shape of the exhaust manifold and several domains (exhaust gas-solid -ambient) require to use the whole assembly meshing. The assembly meshing refers to mesh the whole model as a single mesh process, as compared to part or body-based meshing, in which meshing occurs in the part or body level respectively. They are two algorithms are available for the element type of meshing: CutCell and Tetrahedrons. The CutCell method which is used in this study. It is a Cartesian meshing, which is the general-purpose meshing method designed for ANSYS (Fluent). The solid manifold, fluid volume in the exhaust manifold and ambient are determined as a computational domain. In the present work, the exhaust manifold used in the CFD analysis has been divided into 3668237elements and 3844322 nodes for mesh generation as shown in Fig. 5a, b.

![Figure 4. The exhaust manifold: (a) Solidworks model, (b) original model.](image)

The process begins with a coarse mesh and regularly refining it until the variations noticed in the results are smaller than a predetermined acceptable error. The acceptable error is specified by taking numerical accuracy and the time consuming into consideration. Table 3 and Fig.6, explains the mesh dependency for an arbitrary case within the study area.

![Figure 5a, b. The assembly meshing (CutCell element method).](image)

4.1 Grid Independent Test:

Before starting to generate the CFD production runs, choosing the right mesh type affects the accuracy of the solution and the speed of reaching the results. The mesh independence study was carried out for the exhaust manifold. The aim behind this is to find out the best mesh properties for an accurate solution. The process begins with a coarse mesh and regularly refining it until the variations noticed in the results are smaller than a predetermined acceptable error. The acceptable error is specified by taking numerical accuracy and the time consuming into consideration. Table 3 and Fig.6, explains the mesh dependency for an arbitrary case within the study area.
4.2 Material Selection:

The entities in this research required the material properties to be specified classified as fluid and solid. Flue gas is selected as the working fluid within the exhaust manifold. Flue gas is a mixture of combustion products, fuel, and air. Table 4 list the content equations to calculate the properties of flue gas depending on temperature. The solid material selected for the exhaust manifold is a gray cast iron were Table 5 lists the properties of the solid material. The experimental values of the boundary conditions are mass flow rate and temperature at the inlet surface and static pressure at the outlet surface was set in ANSYS R19.

Table 3. The grid independent tests.

| mesh | Number of element | T_gas °C (upper pipe) | T_gas °C (lower pipe) | Heat flux (w) |
|------|-------------------|-----------------------|-----------------------|--------------|
| 1    | 1478938           | 596.65                | 582.96                | -520.56      |
| 2    | 2175894           | 590.343               | 575.23                | -576.44      |
| 3    | 2944875           | 587.83                | 573.88                | -598.23      |
| 4    | 3668237           | 587.282               | 573.133               | -601.57      |
| 5    | 4698455           | 587.155               | 573.102               | -602.34      |

![Figure 6](image_url) The variation of exhaust gas temperature of the upper pipe versus with total element number.

4.3 Turbulence Model:

The choice of turbulence model depends on considerations such as the physics encompassed in the flow, the established practice for a specific class of problem, the level of accuracy required, the available computational resources, and the amount of time available for the simulation. The chosen of a suitable turbulent model affects the accuracy of numerical results. Three turbulent models, a

| Propriety                     | Equation                                      |
|------------------------------|-----------------------------------------------|
| Density ρ (kg/m³)            | $\frac{353}{T_g}$                             |
| Viscosity μ (pas)            | $1.384 \times 10^{-5} + 2.68 \times 10^{-8} T_g$ |
| Thermal conductivity K (w/k.m) | $8.459 \times 10^{-3} + 5.7 \times 10^{-5} T_g$ |
| Specific heat C_p            | $962.097 + 0.1507 T_g$                        |

$T_g$: exhaust gas temperature in kelvin

| Autoref 3. The equation of gas properties [8]                                      | Table 5. Gray cast iron properties [9]                              |
|-------------------------------------------------------------------------------|---------------------------------------------------------------------|
| Density [kg/m³]                                                               | 7200                                                                |
| Specific heat [J/kg-k]                                                        | 510                                                                 |
| Thermal conductivity[W/m-k]                                                   | 45                                                                  |

|
realizable k-epsilon model, the standard k-ω model, and the SST k-ω model were used frequently in engineering applications to investigate the steady-state, three-dimensional turbulent flow because these models provide a good compromise between computational time and accuracy [10]. To further verify the best turbulent model for the present case, four turbulent models were simulated with the same actual exhaust manifold dimensions, and the results were compared with the experimental data of the same exhaust manifold dimensions, as shown in Table 6. The results showed that the k-ω model is most similar to the experimental data.

Table 6. The test turbulent models.

| No | Turbulence model | T_gas °C outlet (upper pipe) | T_gas °C outlet (lower pipe) |
|----|------------------|-------------------------------|-----------------------------|
| 1  | k-ω              | 587.28                        | 573.1                       |
| 2  | SST k-ω          | 584.45                        | 570.32                      |
| 3  | k-ε              | 578.33                        | 565.57                      |
| 4  | k-ε RNG          | 578.89                        | 566.09                      |
| 5  | Experimental     | 610                           | 602                         |

4.4 Governing Equations of the Numerical Solution:

The governing equations solved in the flow field are the continuity (mass conservation), The Navier-Stokes equations of motion (momentum conservation) and energy equations in three dimensions for the fluid domain (flue gas). The set of the cylindrical differential form of equations describing the transport of mass, momentum, and energy in a fluid under steady-state condition is described as follows; [11]

The Mass conservation (Continuity equation):

\[
\frac{\partial \rho}{\partial t} + \frac{1}{r} \frac{\partial}{\partial r} (\rho rv_r) + \frac{1}{r} \frac{\partial}{\partial \theta} (\rho v_\theta) + \frac{\partial}{\partial z} (\rho v_z) = 0
\]  

The Momentum Conservation equation (Navier-Stokes Equation) at the cylindrical coordinates:

At the r-direction:

\[
\rho \left( v_r \frac{\partial v_r}{\partial r} + v_\theta \frac{v_r}{r} \frac{\partial v_r}{\partial \theta} - \frac{v_z^2}{r} + v_z \frac{\partial v_z}{\partial z} \right) = \rho g_r - \frac{\partial p}{\partial r} + \mu \left[ \frac{\partial}{\partial r} \left( \frac{1}{r} \frac{\partial (rv_r)}{\partial r} \right) + \frac{1}{r^2} \frac{\partial^2 v_r}{\partial \theta^2} - \frac{2}{r^2} \frac{\partial v_\theta}{\partial \theta} + \frac{\partial^2 v_z}{\partial z^2} \right]
\]  

At the θ-direction:
\[
\rho \left( v_r \frac{\partial v_\theta}{\partial r} + \frac{v_\theta}{r} \frac{\partial v_r}{\partial \theta} + v_z \frac{\partial v_\theta}{\partial z} \right) = \rho g_\theta - \frac{1}{r} \frac{\partial p}{\partial \theta} + \mu \left[ \frac{\partial}{\partial r} \left( \frac{1}{r} \frac{\partial (rv_\theta)}{\partial r} \right) + \frac{1}{r^2} \frac{\partial^2 v_\theta}{\partial \theta^2} + \frac{2}{r^2} \frac{\partial v_r}{\partial \theta} + \frac{\partial^2 v_\theta}{\partial z^2} \right]
\] (7)

At the Z-direction:
\[
\rho \left( \frac{\partial v_z}{\partial t} + v_r \frac{\partial v_z}{\partial r} + \frac{v_\theta}{r} \frac{\partial v_z}{\partial \theta} + v_z \frac{\partial v_z}{\partial z} \right) = \rho g_z - \frac{\partial p}{\partial z} + \mu \left[ \frac{1}{r} \frac{\partial}{\partial r} \left( \frac{1}{r^2} \frac{\partial (rv_z)}{\partial r} \right) + \frac{1}{r^2} \frac{\partial^2 v_z}{\partial \theta^2} + \frac{\partial^2 v_z}{\partial z^2} \right]
\] (8)

Energy Equation:
\[
\rho C_p \left( \frac{\partial T}{\partial t} + v_r \frac{\partial T}{\partial r} + \frac{v_\theta}{r} \frac{\partial T}{\partial \theta} + v_z \frac{\partial T}{\partial z} \right) = k \left[ \frac{1}{r} \frac{\partial}{\partial r} \left( \frac{1}{r^2} \frac{\partial T}{\partial r} \right) + \frac{1}{r^2} \frac{\partial^2 T}{\partial \theta^2} + \frac{\partial^2 T}{\partial z^2} \right] + \mu
\] (9)

The governing equation of the solid domain (exhaust manifold) [12]
\[
\frac{1}{r} \frac{\partial}{\partial r} \left( kr \frac{\partial T}{\partial r} \right) + \frac{1}{r^2} \frac{\partial}{\partial \theta} \left( kr \frac{\partial T}{\partial \theta} \right) + \frac{\partial}{\partial z} \left( k \frac{\partial T}{\partial z} \right) + \dot{g} = 0
\] (10)

5. Results and Discussions:

5.1 Experimental Results:

Testing was conducted on the experimental platform to evaluate all the operating conditions of the engine. The testing employed by choosing four spanning speeds of the engine (1800-3000) with a step increase of 400 rpm, and related to three different loads per speed. Loads are precisely selected to harmonize the work of the engine, where each speed is determined with a maximum load that cannot be exceeded. Also, high loads can cause damage to the thermocouples, especially those that measure exhaust gas temperatures. The range of loads used in this testing that stretched from (12-67) N.m are detailed in Table 2. Figs. (7-10) appear the range of skin temperatures along the exhaust manifold for different engine speeds, 1800, 2200, 2600 and 3000 rpm under low, middle and high loading. Skin temperatures ranged from 342 °C (TC-8) to 424 °C (TC-9) for low and high loads respectively at 1800 rpm speed. While, for 3000 rpm speed, the skin temperatures ranged from 452 °C (TC-11) to 500 °C (TC-9) for low and high loads respectively. It is evident, that the lowest surface temperature was recorded at TC-8 for two speeds (1800 and 2200), while at speeds (2600 and 3000) the lowest surface temperature is presented at TC-11. In general, the maximum thermal load represented by high temperatures is located in the manifold area between TC-10 and TC-9 from the top and TC-12 from the bottom for all engine speed. This area is located in the middle of the exhaust manifold making the meeting location of the exhaust gases coming from the first, second and third cylinder ports. In addition, the local skin temperatures and surface heat fluxes increased with increasing engine speed and increasing load.
5.2 Numerical Results

A 3D steady-state CFD simulation for the exhaust manifold was performed under the operating conditions and physical properties as mentioned. Since the greatest material properties and heat transfer coefficients are temperature contingent and also the exhaust gas temperature and component surface temperature is coupled. The thermal solution for each control volume is obtained by an iterative procedure. The initial value of the exhaust gas temperature and other parameters at the first control volume are based on comprehensive boundary conditions such as mass flow rate and inlet gas temperature. The temperature distribution is gained by solving the equations (1-6) repeated until convergence was completed. Fig. 11 shows the numerical simulation results of the skin temperature distribution on the 3D model of the exhaust manifold at engine speed 2200 rpm under different loading and Fig.12 shows the 3D numerical simulation for the skin temperature distribution as a contour of the exhaust manifold at engine speed 3000 rpm with different loadings. The figures show the effect of the load on an increment of the thermal load on the exhaust manifold under the same engine speed. The temperature difference between the lowest and highest load was about 30 °C at the speed 2200 rpm while the difference contracts to approximately 20 °C at speed 3000 rpm.
The skin temperatures distribution of the upper pipe and lower pipe along the z-axis direction starting from port 2 towards the exit of the exhaust manifold, curves are shown in Figs. (13-16) at two engine speeds with various loading.
Comparison Assessment Between Experimental Analysis and Numerical Analysis:

Figures (17 and 18) show the comparative assessment of the simulation predicting results and measured manifold skin temperatures, according to thermocouple located on the exhaust manifold body and through a wide range of engine operating conditions. Figures (17 and 18) confirmed that the comparative assessment between the simulated skin temperatures at the upper pipe junction TC-9 and the lower pipe junction TC-12 data are very close, which are showing that the experimental measurements are few higher than the numerical predicted data. This is taken into consideration the spatial variations of the exhaust manifold wall temperatures measured by skin thermocouples mounted along with various locations of the manifold. Thus, the model predicts the manifold skin temperatures are reasonably well.

**Figure 13.** Upper pipe skin temperatures vs. distance at 2200 rpm.

**Figure 14.** Lower pipe skin temperatures vs. distance at 2200 rpm.

**Figure 15.** Upper pipe skin temperatures vs. distance at 3000 rpm.

**Figure 16.** Lower pipe skin temperatures vs. distance at 3000 rpm.

5.3 Comparison Assessment Between Experimental Analysis and Numerical Analysis:
6. CONCLUSION

The exhaust manifold of a multi-cylinder engine was experimentally and numerically assisted looking for a better understanding of the heat transfer during the exhaust manifold operation. The assessment study was carried out to compare the experimental and numerical CFD, heat transfer analyses data of the gas flow in/out the exhaust manifold. In the experimental analysis, the temperatures at the inlet and outlet surfaces of the exhaust manifold were measured. The numerical analysis produced a three-dimensional model simulated under steady-state conditions. The properties of the exhaust gas were specified depending on the average temperature in the ANSYS/Fluent version R19. Experimental values of the boundary conditions are mass flow rate and temperature at the inlet surface and static pressure at the outlet surface were set in ANSYS R19. The results showed that the numerical results provide a satisfactory estimation of the fluid flow in the exhaust pipe. Also, the external local skin temperatures varied significantly throughout the surface area of the manifold. The local skin temperatures increased with increasing engine speed and increasing load. The numerical modeling and simulation using ANSYS software presented in the study could be useful for the determination of the temperature distribution along the exhaust manifold for any gasoline engine due to the converge results with the experimental.

7. ACKNOWLEDGMENTS

The authors would like to thank the staff of the combustion laboratory in the Department of Mechanical Engineering at the University of Technology for their serious cooperation for the success of this study.
REFERENCES

[1] C. D. Bannister, C. J. Brace, J. Taylor, T. Brooks, and N. Fraser, “An empirical approach to predicting heat transfer within single- and twin-skin automotive exhaust systems,” Proc. Inst. Mech. Eng. Part J. Automob. Eng., vol. 225, no. 7, pp. 913–929, 2011.

[2] G. C. Mavropoulos, C. D. Rakopoulos, and D. T. Hountalas, “Experimental investigation of instantaneous cyclic heat transfer in the combustion chamber and exhaust manifold of a DI diesel engine under transient operating conditions,” SAE Technical Paper, 2009.

[3] B. Celikten, I. Duman, C. Harman, and S. Eroglu, “Exhaust Manifold Thermal Assessment with Ambient Heat Transfer Coefficient Optimization,” SAE Int. J. Passeng. Cars-Mech. Syst., vol. 11, no. 06-11-0016, pp. 193–202, 2018.

[4] K. Haehndel, T. Frank, F. M. Christel, C. Spengler, G. Suck, and S. Abanteriba, “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., vol. 6, no. 2013-01-0879, pp. 847–858, 2013.

[5] S. Eroglu, I. Duman, A. H. Guzel, and R. Yilmaz, “Durability Analysis of Heavy Duty Engine Exhaust Manifold Using CFD-FE Coupling,” SAE Technical Paper, 2016.

[6] C. Depcik and D. Assanis, “A universal heat transfer correlation for intake and exhaust flows in an spark-ignition internal combustion engine,” SAE Trans., pp. 734–740, 2002.

[7] A. Fluent, “ANSYSVR [ANSYS Fluent]. 15.0, Help System, User’s Guide/Theory Guide,” ANSYS Inc Canonsburg PA HttpwwwAnsysComProductsFluidsANSYS-Fluent, 2017.

[8] M. Durat, Z. Parlak, M. Kapsiz, A. Parlak, and F. Fiçici, “CFD AND EXPERIMENTAL ANALYSIS ON THERMAL PERFORMANCE OF EXHAUST SYSTEM OF A SPARK IGNITION ENGINE.,” Isi BilimiVeTek. DergisiJournal Therm. Sci. Technol., vol. 33, no. 2, 2013.

[9] Y. S. Touloukian and T. Makita, “Thermophysical properties of matter-the TPRC data series. Volume 6. Specific heat-nonmetallic liquids and gases.(Reannouncement). Data book,“ Purdue Univ., Lafayette, IN (United States).Thermophysical and Electronic …, 1970.

[10] F. Menter, “Zonal two equation kw turbulence models for aerodynamic flows,” in 23rd fluid dynamics, plasmadynamics, and lasers conference, 1993, p. 2906.

[11] L. M. Jiji and L. M. Jiji, Heat convection. Springer, 2006.

[12] Y. A. Cengel, S. Klein, and W. Beckman, Heat transfer: a practical approach, vol. 141. McGraw-Hill New York, 1998.
DEFINITIONS/ABBREVIATIONS

$\mu$ - Dynamic Viscosity (Pas)

$\mu_{\text{bulk}}$ - Bulk Fluid Viscosity (Pas)

$\mu_{\text{skin}}$ - Surface Viscosity (Pas)

$C_p$ - Specific Heat (J/kgK)

$k$ - Thermal conductivity (W/mK)

$\rho$ - Density of Fluid (m3/kg)

$T$ - Temperature (°C)

$p$ - pressure (pa)

g - acceleration of grained (m/s$^2$)

$V_r$ - r-velocity (m/s)

$V_\theta$ - $\theta$-velocity (m/s)

$V_z$ - z-velocity (m/s)

CFD - Computational Fluid Dynamic

RPM - Revelation per minute

Fluent - Fluid and heat transfer software