A Mathematical Model for the Exhaust Gas Temperature Profile of a Diesel Engine

You may also be interested in:

A MATHEMATICAL MODEL FOR PREDICTING NIGHT-SKY
Mark A. Yocke, Henry Hogo and Don Henderson

Mathematical modeling of piezoresistive elements
M Geremias, R C Moreira, L A Rasia et al.

Mathematical modeling of the instability of viscous fluid films
L.A. Prokudina

Mathematical modelling to predict the roughness average in micro milling process
C Burlacu and O Iordan

A mathematical model of the vertical dual-mass hydroimpulsive mechanism
F A Simankin, G R Ziyakaev, I V Kuznetsov et al.

Development of Requirements for a Basic Standardized Mathematical Model of Geokhod
V V Aksenov, V Yu Beglyakov, A A Kazantsev et al.

Mathematical model and software for control of commissioning blast furnace
N A Spirin, O P Onorin, K A Shchipanov et al.

A Mathematical Model for the Exhaust Gas Temperature Profile of a Diesel Engine

C H G Brito[1]**, C B Maia**[2] **and J R Sodré**[3]

Department of Mechanical Engineering, Pontifical Catholic University of Minas Gerais, Av. Dom José Gaspar, 500, Belo Horizonte, MG, 30531-910, Brazil

E-mail: cristianohbrito@globo.com[1]; cristiana@pucminas.br[2]; ricardo@pucminas.br[3]

Abstract. This work presents a heat transfer model for the exhaust gas of a diesel power generator to determine the gas temperature profile in the exhaust pipe. The numerical methodology to solve the mathematical model was developed using a finite difference method approach for energy equation resolution and determination of temperature profiles considering turbulent fluid flow and variable fluid properties. The simulation was carried out for engine operation under loads from 0 kW to 40 kW. The model was compared with results obtained using the multidimensional Ansys CFX software, which was applied to solve the governor equations of turbulent fluid flow. The results for the temperature profiles in the exhaust pipe show a good proximity between the mathematical model developed and the multidimensional software.

1. Introduction

There is a growing need to produce more efficient internal combustion engines, since the laws that regulate the level of pollutant emissions and fuel consumption by automobiles are increasingly demanding lower limits. Together with regulatory laws, government incentive laws also encourage the manufacturers to continuously develop new technologies that allow automobiles to reach and even exceed the consumption limits and emissions. Heat transfer is an important requirement in the analysis, development and design of internal combustion engine, being directly related to engine efficiency and emissions levels [1]. Within the various lines of research and approaches for heat transfer analysis at internal combustion engines, a special focus is given to the exhaust system.

The equations governing the fluid flow and determining the temperature profiles, when analyzed in the plane or space, results in complex calculations and hard analytical solution. This problem can be mitigated with the adoption of numerical methods, such as finite difference method (FDM) and finite volume method (FVM). FDM is a numerical method generally used to study heat transfer and fluid flow dynamics [2]. In the analysis of exhaust systems, the FDM has been employed to verify the energy availability of exhaust gases [3-7]. The exhaust gas fluid flow and heat transfer was simulated via FDM to design optimization of the exhaust system [8].

The main objective of this work is to evaluate the consistency of heat transfer solution from exhaust pipe modeling on FDM from comparison with results obtained in computational fluid dynamics (CFD) modeling using Ansys CFX software. The FDM modeling was adopted for energy equation resolution and determination of the exhaust gas temperature profiles, considering turbulent fluid flow and variable properties. The numerical simulation was carried out for engine operation under loads from 0 kW to 40 kW.

2. Problem formulation

The exhaust system is responsible by about 30% of the energy loss from an internal combustion engine [9]. The energy carried by the exhaust gas can be evaluated from the analysis of the internal fluid flow. The exhaust pipe taken as the physical domain of this work is shown in Fig. 1. The heat transfer and fluid flow is analyzed taking the duct as a straight pipe, as shown by Fig. 2.
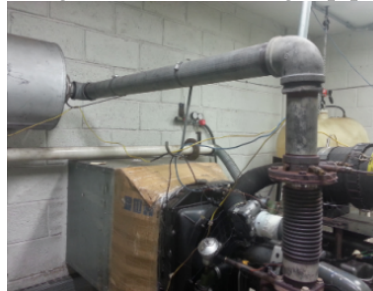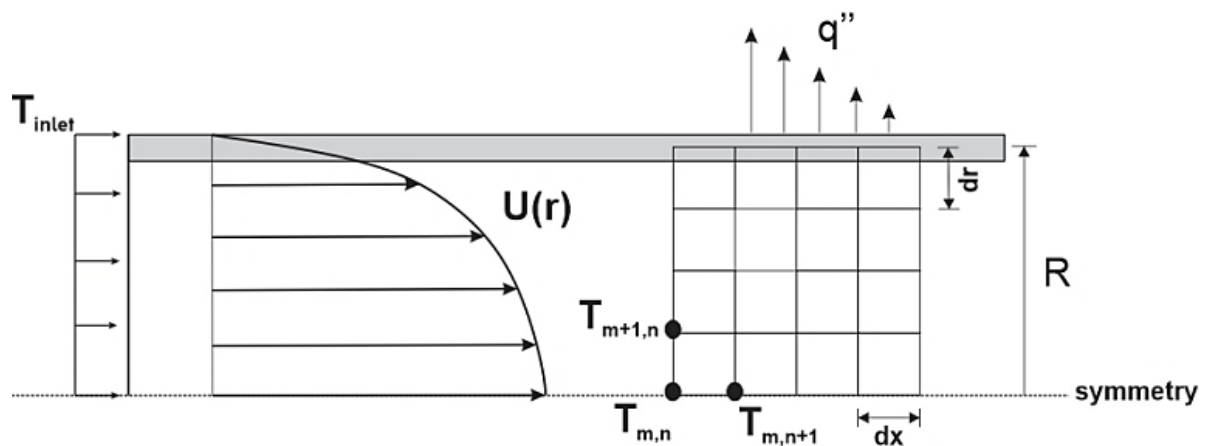

Figure 1. Exhaust pipe

Figure 2. Nodal mesh

3. Mathematical model

In order to determine the temperature profiles it has been proposed a model based on FDM. The implemented model solve the energy equation from fundamental assumptions: (1) steady state; (2) fully developed turbulent flow; (3) turbulent velocity profile 1/7; (4) variable properties; (5) fluid modelled as air.

The results of this model are compared with results obtained from the Ansys CFX® software, in which the governing equations of turbulent flow are resolved. The governing equations are the continuity equation (Eq. 1), momentum equation (Eq. 2) and energy equation (Eq. 3) [10]:

$$\frac{\partial}{\partial t}\rho + \nabla \bullet \left(\rho \overline{\mathbf{U}}\right) = 0 \tag{1}$$

$$\frac{D\left(\rho \overline{\mathbf{U}}\right)}{Dt} = \rho \mathbf{g} - \nabla \overline{p} + \nabla \bullet \tau_{ij} \tag{2}$$

$$\rho c_p \frac{D\overline{T}}{Dt} = -\frac{\partial}{\partial x_i}q_i + \overline{\Phi} \tag{3}$$

The K-ω SST model [11] is used to treat the turbulence regime in the present study in the simulation with Ansys CFX®.

4. Numerical methodology

The use of numerical techniques to solve engineering problems arises from the difficulty to achieve analytical solutions to highly complex problems, which involve partial differential equations and implied equations, among others. The currently available computational speed enables the effectiveness of the solutions for such problems [2]. In this research two numerical methods were used to solve the governing equations of the exhaust gas flow. The FDM was used to solve the energy equation and FVM was used to solve the seven governing equations of the exhaust gas flow.

4.1. FDM Model

The discretization of the energy equation, based on FDM developed using MATLAB® software (Fig. 3), was done by the bidimensional nodal mesh shown by Fig. 2. The boundary conditions are [12]:

$$\frac{d\theta}{dr} = \begin{cases} 0 & \to r = 0 \\ -\text{Bi} \cdot \theta & \to r = R \end{cases} \quad \text{where} \quad \theta = \frac{T_\infty - T(r,x)}{T_\infty - T_m(x)} \tag{4}$$

For turbulent, fully developed flow the 1/7 velocity profile is able to satisfactorily approximate the behaviour of the real flow velocity profiles [10].

To determine the equations of the fluid intensive properties as a function of the exhaust gas temperature, modelled as air, it was used the Engineering Equation Solver (EES®) software.
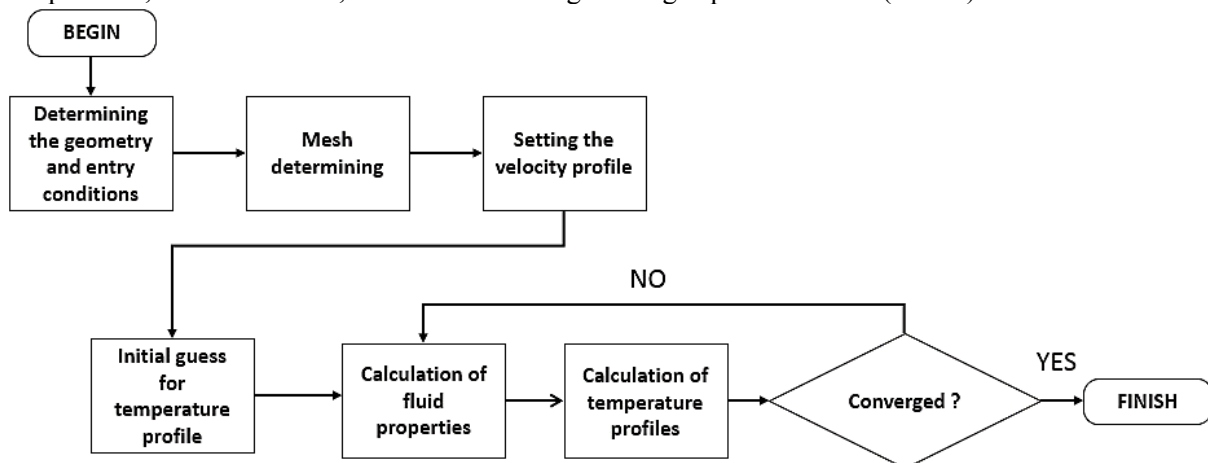
Figure 3. Algorithm flowchart of FDM implemented using MATLAB® software.

4.2. FVM Model

The FVM model was solved using Ansys CFX® 14.5 software, adopting a straight pipe geometry and considering symmetry and 3-dimensional fluid dynamics and heat transfer. This model was solved to compare with the FDM model. The fundamental hypotheses adopted are: (1) air at 25°C, (2) turbulent flow, (3) steady state condition, and (4) incompressible flow. The entry conditions are: (1) pipe outlet with relative static pressure of 0 Pa, (2) non-sliding walls conditions, (3) known experimental mass flow rate and flow temperature, (4) prescribed heat transfer coefficient in the pipe wall, and (5) ambient temperature 27°C.

4.3. Entry conditions of numerical models

In the numerical simulations it was adopted input and boundary conditions obtained experimentally [13] and described in Table 1. These data were collected from a diesel power generator operating under loads of 0, 10, 20 and 40 kW.

Table 1. Data entry conditions used in the numerical simulations.

Nominal power	kW	0	10	20	40
Ambient temperature	°C	27	27	27	27
Exhaust gas inlet temperature	°C	149.4	239.37	332.12	598.27
Exhaust gas mass flow rate	kg/h	148.4	144.71	142.72	132.95

5. Results and discussion

In the algorithm implemented in Matlab software it was adopted square mesh to ensure compliance with the proposed mesh stability criterion. The convergence criterion adopted the value of 1% [2]. Mesh with 0.17 mm element size was adopted for the simulation. The FVM model in Ansys CFX® 14.5 software was simulated with 11.6 million elements. The numerical solutions proposed to the problem of exhaust gas flow are here evaluated in a comparative way. For such it is evaluated the temperature profiles obtained in the section 150 mm before the pipe outlet. The FDM model was named Matlab-FDM and FVM model was named Ansys-SP. The temperature profiles in Fig. 4 and Fig. 5 show that the Matlab-MDF model depicts a behaviour similar to Ansys-DR model and the temperature gradients, increasing as the engine load power is increased.
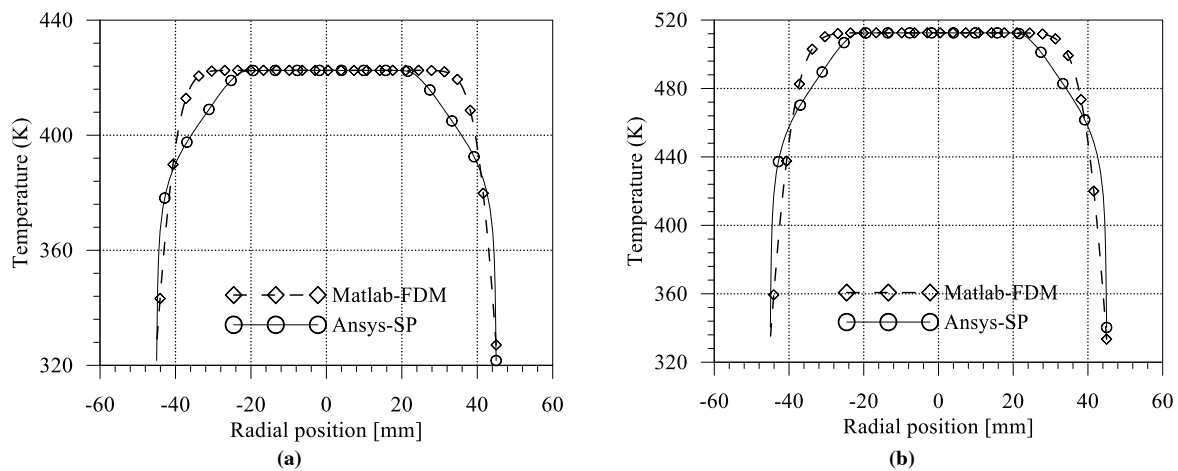

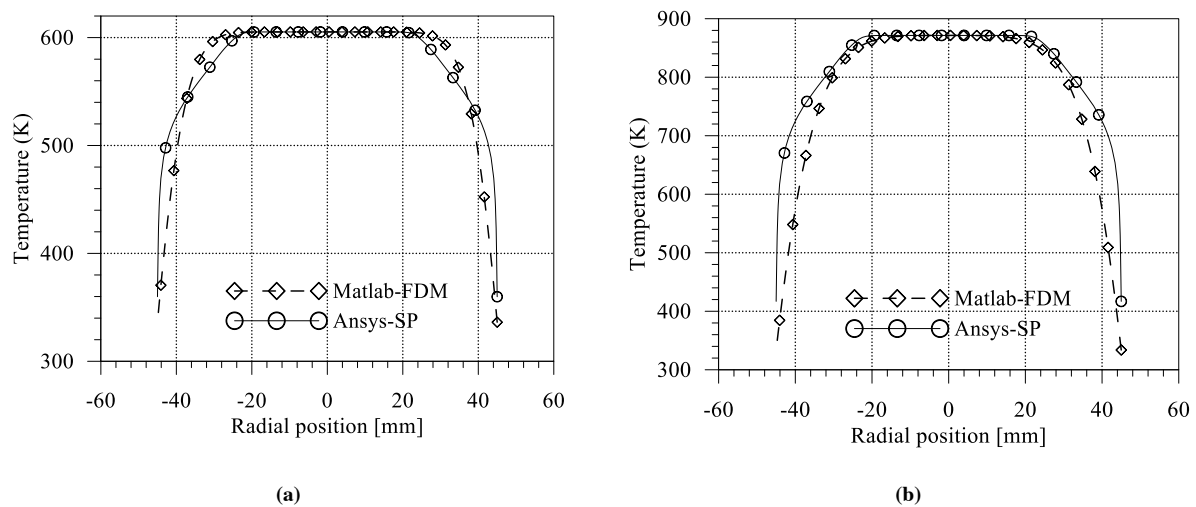**Figure 4.** Temperature profiles for engine loads: (a) 0 and (b) 10 kW

Figure 5. Temperature profiles for engine loads: (a) 20 and (b) 40 kW

6. Conclusions

With the numerical methodology used, it was possible to determine the temperature profiles in the exhaust pipe of a stationary engine operating under different loads. The results show a good correlation between the proposed Matlab-FDM model and the Ansys-SP model. Temperature profiles obtained in both cases show that, as the load applied to the engine is increased, the temperature gradient along the pipe length is also increased as a result of increased exhaust gas temperature and reduced mass flow rate.

References

[1]   Drenth A C, Olsen D B, Cabot P E and Johnson J J 2014 Compression ignition engine performance and emission evaluation of industrial oilseed feedstock camelina, carinata and pennycress across three fuel pathways *Fuel* **136** 143-155

[2]   Özisik N (1994) *Finite Difference Methods in Heat Transfer* (Florida : CRC Press)

[3]   Rakopoulos C, Andritsakis E, Hountalas D 1995 The influence of exhaust system unsteady gas flow and insulation on the performance of turbocharged diesel *Heat Recovery Systems & CHP* **15** 51-72

[4]   Baker C, Vuppuluri P, Shi L and Hall M 2012 Model of Heat Exchanger for Waste Heat Recovery from Diesel Engine Exhaust for Thermoelectric *Power Generation Journal of Electronic Materials* **41** 1290-1297

[5]   Moazami N, Wyszynski M L and Mahmoudi H 2013 Modeling of catalytic monolith reactor for reforming of hexadecane with exhaust gas *International Journal of Hydrogen Energy* **38** 11826-11839

[6]   Galindo J, Serrano J R, Piqueras P, García-Afonso O 2012 Heat transfer modelling in honeycomb wall-flow diesel particulate filters *Energy* **43** 201-213

[7]   Brito C H G, Andrade G F and Sodré J R 2014 Análise do Perfil de Temperaturas no Duto de Exaustão de um Motor pelo Método das Diferenças Finitas *Proceeding Series of the Brazilian Society of Computational and Applied Mathematics* **2** 10511-10516

5

[8]   Kesgin U 2005 Study on the design of inlet and exhaust system of a stationary internal combustion engine *Energy Conversion and Management* **46** 2258-2287

[9]   Pulkrabek W W 1997 *Engineering Fundamentals of the Internal Combustion Engine* (New Jersey: Pratice Hall) pp 312

[10]   White F (2007) *Fluid Mechanics* 6. Ed. (New York : McGraw-Hill)

[11]   Versteeg H K and Malalasekera W (2007) *An Introduction to Computational Fluid Dynamics: The Finite Volume Method* 2. Ed. (London: Pearson Education)

[12]   Bejan A (2012) *Convective Heat Transfer* 2. Ed. (New Jersey: John Wiley & Sons)

[13]   Morais A et al 2013 Hydrogen impacts on performance and CO2 emissions from a diesel power generator *Internal Journal Hydrogen Energy* **38** 6857-6864