BULETINUL INSTITUTULUI POLITEHNIC DIN IAŞI
Publicat de
Universitatea Tehnică „Gheorghe Asachi” din Iaşi
Tomul LXI (LXV), Fasc. 2, 2015
Secţia
CONSTRUCŢII DE MAŞINI

NUMERICAL SIMULATION ON EXTERNAL SURFACE CONVECTIVE HEAT TRANSFER IN STATIONARY DIESEL ENGINES

BY

LIVIU COSTIUC[1*], VENEŢIA SANDU[1], VIRGIL B. UNGUREANU[1], VIOREL POPA[2] and MIRCEA IVĂNOIU[1]

[1]Transilvania University, România,
Department of Mechanical Engineering
[2]Dunărea de Jos University, România,
Department of Thermal Systems

Received: July 16, 2015
Accepted for publication: October 8, 2015

Abstract. The aim of this paper is the numerical simulation of heat transfer over the external surface of a stationary Diesel engine to obtain a relation for convective heat flow transferred to environment at different air velocities. The simulation was performed using COMSOL Multiphysics software for forced convection with turbulence. The simulation is using conjugate heat transfer and fluid flow on Turbulent Fluid-Thermal Interaction module. Temperature field used for initial external surface temperature was obtained by IR thermovision measurements on existing stationary engine. Simulations were shown the inlet velocity influence on overall heat transfer coefficient is significant at bigger values of 10 m/s.

Key words: heat transfer coefficient; convection; conduction; turbulence; numerical simulation; Diesel engine.

---

[*]Corresponding author; *e-mail*: lcostiuc@unitbv.ro

1. Introduction

The study of Diesel engine energy balance can improve engine efficiency by reducing energy losses. The chemical energy density of the fuel is converted into effective mechanical power, and a heat flux loss in coolant and oil as cooling fluids, a heat flux loss in exhaust gas and the heat flux loss by heat transfer of the engine surface with external air. This last term is typically estimated with data given in literature by (Heywood, 1988; Martyr & Plint, 2007; Abedin *et al*., 2013; Sandu *et al*., 2014).

The contribution to the heat flux loss by heat transfer is the convective heat loss of the engine in the environment. Because of few literature data regarding convective heat loss through external engine surfaces, the present paper aims are to investigate different heat transfer models by numerical modelling.

The numeric model was developed using COMSOL Multiphysics software (2008) in a forced turbulent convection case. The model is considered as 2D steady-state surface temperature and with influence of air-flow fields using the equations for Turbulent Fluid-Thermal Interaction and numerical modules used in simulations are the Heat Transfer Module and k-ε Turbulence Model.

The aim of the work is to find best values for heat transfer coefficient taking into account the velocity of the air which flows over the engine external surface.

2. Engine Numerical Modelling

2.1. Setup Conditions of the Model

2.1.1. *Temperature*. The ambient conditions were considered as: barometric pressure 718 mmHg and average air temperature of 16ºC. As mentioned by (Sandu *et al*., 2014) the engine surface temperature was measured both locally with thermocouple placed on different points on the engine surface and superficially using a thermography IR camera. The surface temperature values were recorded for three operation modes at rotational speed of 1246 rpm (mode I), 1596 rpm (mode II) and 1354 rpm (mode III), and are presented in Table 1 and Table 2.

Table 1
Parameters of Engine Operation Modes

Operation mode	I	II	III
Speed, rpm	1246	1596	1354
Brake torque, Nm	73.6	122.6	132.5
Volumetric intake air flow rate, m^3/h	104	116	108
Ambient temperature, °C	16.0	16.0	16.0
Engine temperature (thermocouple), °C	60	62	64

Table 2
IR Engine Mean Temperatures in °C

Mean temperature/Mode	I	II	III
Right view	59.7	64.3	68.4
Left view	71.9	79.0	81.3
Upper (bottom) view	42.8	54.7	60.0
Front view	59.2	61.5	63.2
End view	60.6	63.2	69.9
Overall surface mean temperature	60.9	66.4	69.5

The mean surface temperatures are used in simulations as body temperature of the model.

2.1.2. *Air velocity*. The air surrounding the engine has a motion due to fan operation so the air flows over the engine surfaces. The fan is mounted directly on the crankshaft and fan speed is the same with engine speed. The air velocity distribution around the engine was taken in five points represented in Fig. 1. As mentioned in (Sandu *et al*., 2014) the inlet air flow velocity is: 6.9 m/s for operation mode I, 12.8 m/s for operation mode II and 8.7 m/s for operation mode III. So, the values considered in present simulation as inlet velocities of the air are 1, 5 10 and 15 m/s to cover the range of above mentioned values.

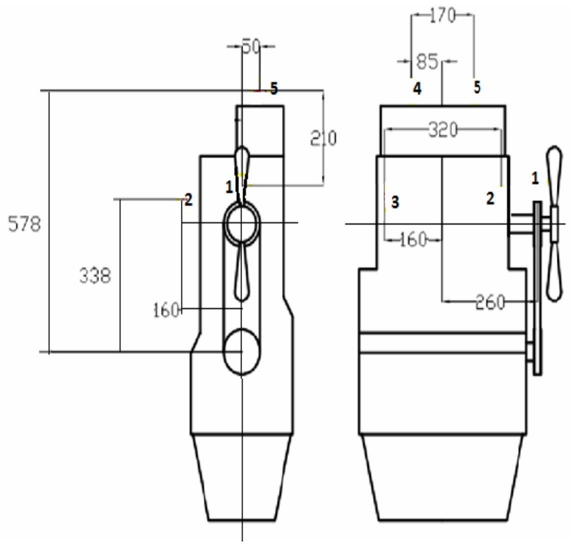

Fig. 1 − Positions of measurement points around the engine.

2.1.3. *Geometry-Mesh*. The 2D heat transfer and air flow model is considered in a horizontal plane of the engine because the heat transfer coefficient is investigated on the left and right side of the engine. Taking into account all surfaces supposes a 3D modelling which will be considered further. The engine section with vertical symmetry is 0.200 m width and 0.440 m long. The discretization of air and body engine domain using triangular elements is shown in Fig. 2, and is resulting 26771 degrees of freedom and 3898 triangular elements.

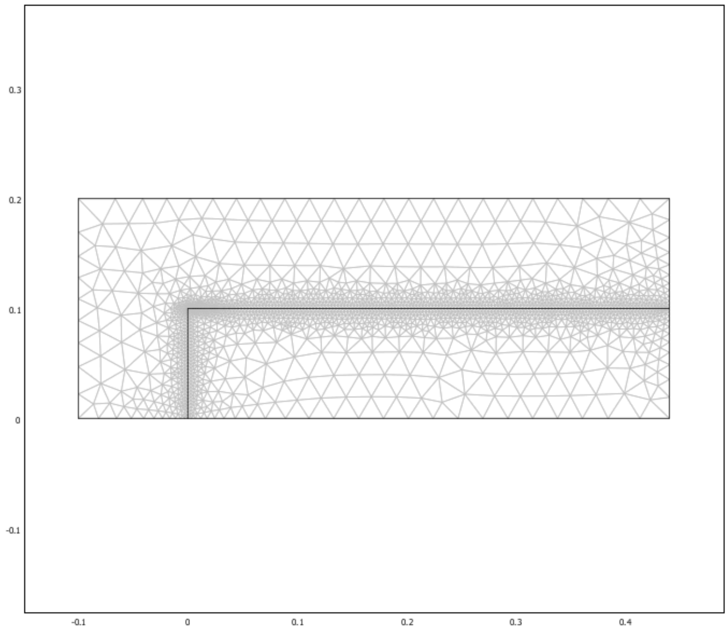

Fig. 2 − Mesh geometry.

2.1.4. *Boundary mode*. The boundary for air and body engine domain is shown in Fig. 3, and the boundary type settings on each study domain are presented in Table 3.

Table 3
Boundary Settings

Domain 1 Air	Boundary	1	2-3	4, 6	8
	Type	Temperature	Insulation/ Symmetry	Thermal wall function	Convective flux
Domain 2 Engine body	Boundary	4, 6	5	7	
	Type	Heat flux	Temperature	Insulation/ Symmetry	

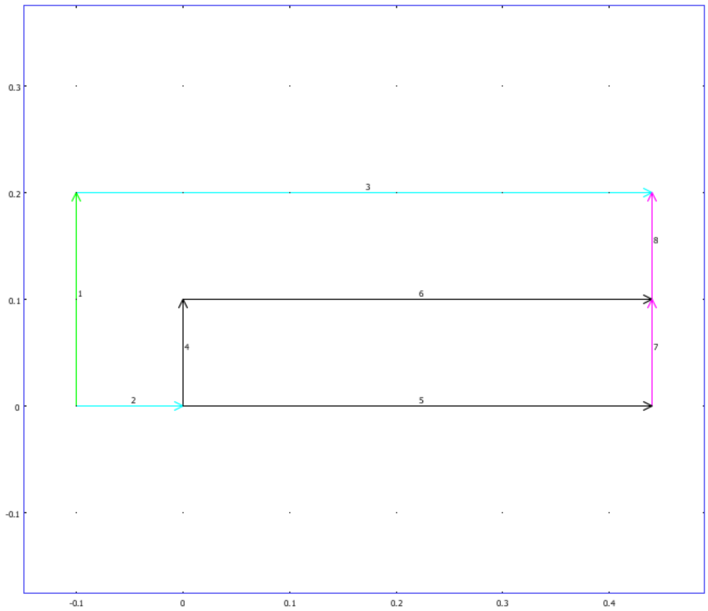

Fig. 2 – Boundary geometry.

2.2. Mathematical Model

The model uses the Reynolds-averaged Navier-Stokes (RANS) equations in the air domain and conductive and convective heat equation in the air and the solid wall. The properties for the fluid are the air at atmospheric pressure, and for the solid those of cast iron. The air properties are considered as temperature dependent. The equations for air domain, which includes equations for turbulent kinetic energy k and for dissipation ε, are described as:

$$\nabla \cdot (\rho u) = 0 \tag{1}$$

$$\rho(u \cdot \nabla) \cdot u = \nabla \cdot \left[-p \cdot I + (\eta + \eta_T)\left(\nabla u + (\nabla u)^T - \frac{2}{3}(\nabla \cdot u) \cdot I \right) - \frac{2}{3}\rho \cdot k \cdot I \right] \tag{2}$$

$$\rho u \cdot \nabla k = \nabla \cdot \left[\left(\eta + \frac{\eta_T}{\sigma_k} \right) \nabla k \right] + \eta_T \cdot P(u) - \left(\frac{2}{3}\rho \cdot k \right) \nabla \cdot u - \rho \varepsilon . \tag{3}$$

$$\rho u \cdot \nabla \varepsilon = \nabla \cdot \left[\left(\eta + \frac{\eta_T}{\sigma_\varepsilon} \right) \nabla \varepsilon \right] + \left(C_{\varepsilon 1} \cdot \frac{\varepsilon}{k} \right) \left[\eta_T \cdot P(u) - \left(\frac{2}{3}\rho \cdot k \right) \nabla \cdot u \right] - C_{\varepsilon 2} \cdot \rho \frac{\varepsilon^2}{k} \tag{4}$$

$$P(u) = \nabla u : \left(\nabla u + (\nabla u)^T\right) - \frac{2}{3}(\nabla \cdot u)^2; \quad \eta_T = \rho \cdot C_\mu \cdot \frac{k^2}{\varepsilon}. \tag{5}$$

The turbulence is considered so it is necessary to compute the fluid thermal conductivity taking into account the effect of mixing due to eddies. The effective thermal conductivity is calculated according to the eq. (6).

$$k_{eff} = k + k_T = k + C_P \cdot \eta_T. \tag{6}$$

where k is the physical thermal conductivity of the fluid, k_T is the turbulent conductivity, η_T denotes the turbulent viscosity and C_p equals the heat capacity.

The boundary conditions for this application used are the two-equation turbulence models such as the k-ε model for fluid domain and the heat transport equations for solid. The k-ε equations in the fluid domain considers the specified velocity at the inlet boundary, specified pressure at the outlet boundary, symmetry at the top and bottom boundary and logarithmic wall function at the solid surface boundaries. The heat transport equations are solved using room temperature at the inlet, convection-dominated transport at the outlet, thermal wall function at the solid/air interface. For the solid-fluid interfaces the model uses the logarithmic wall function boundary condition for turbulent flow.

The heat flux q, [W/m^2] is calculated from equation:

$$q = \rho \cdot C_p \cdot C_\mu^{0.25} \cdot k_w^{0.5} \cdot (T_w - T)/T^+. \tag{7}$$

where ρ and C_p are the fluid's density and heat capacity, respectively, C_μ is a numerical constant of the turbulence model and k_w is the value of the turbulent kinematic energy at the wall, T_w equals to the temperature of the solid at the wall, while T is the fluid temperature.

The dimensionless quantity T^+ is related to the dimensionless wall offset, δ_w^+, through the definition: $T^+ = \frac{\mathrm{Pr}_T}{\kappa}\ln(\delta_w^+) + \beta$, where the turbulent Prandtl number Pr_T is fixed to 1.0, the von Karman constant κ is set to 0.42, and β model constant is set to 3.27. The dimensionless wall offset is defined as $\delta_w^+ = \rho \cdot \delta_w \cdot C_\mu^{0.25} k_w^{0.5} / \eta$, where δ_w is the specified wall offset.

Other model constants used are $C_{\varepsilon 1}$=1.44, $C_{\varepsilon 2}$=1.92, σ_k=1.0, σ_ε=1.3, C_μ=0.09.

3. Results and Discussions

After the simulation in the post-processing section of the software the only temperature colour profile plot for u = 15 m/s have been chosen. This plot is presented in Fig. 4. Also, the Reynolds number and Nusselt number are plotted for four velocity values in Figs. 5 and 6.

The local heat transfer coefficient plot is presented in Fig. 7. The overall heat transfer coefficient variation in function of inlet velocity is plotted in Fig. 8.

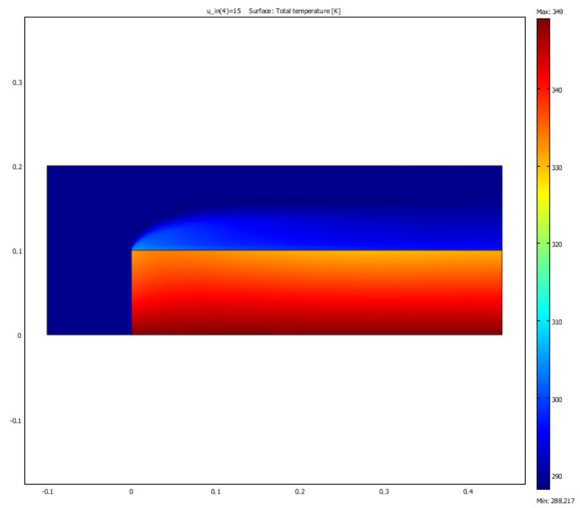

Fig. 4 – Temperature plot for inlet velocity of 15 m/s.

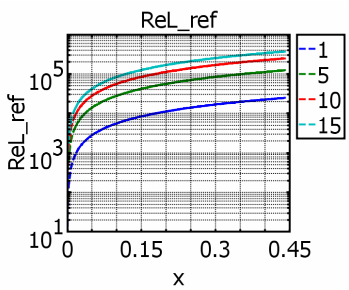

Fig. 5 – Reynolds number for air domain along the engine length for different inlet air velocities.

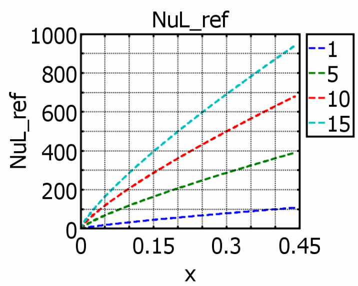

Fig. 6 – Nusselt number on the air domain along the engine length for different inlet air velocities.

The calculations were performed for inlet velocity of 1, 5, 10 and 15 m/s, to be evidenced the influence on heat transfer and local heat transfer coefficient. The surface temperature is set to 66°C. The temperature profile for u = 15 m/s as is shown in Fig. 4 depicts the variations of temperature near to side wall as is expected. To get a point of view of the heat transfer the Reynolds number (Fig. 6) increases rapidly near to solid wall to turbulent flow, so the laminar flow approach for heat transfer coefficient is underestimating method even for inlet velocity is u = 1.0 m/s. The Nusselt number (Fig. 6) is increasing with the increasing of velocity and have a constant increasing along the solid wall at constant velocity.

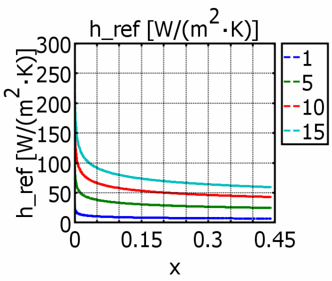

Fig. 7 − Local heat transfer coefficient along the engine length for different inlet air velocities.

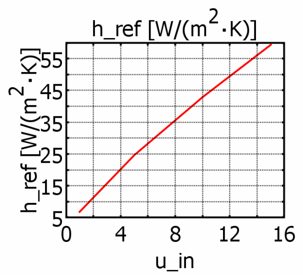

Fig. 8 − Overall heat transfer coefficient in function of inlet air velocity.

The local heat transfer coefficient decreases along the side wall *i.e.* from 120 to 45 W/m^2.K for inlet velocity u = 10 m/s (Fig. 7), and increases as the velocity is increased as is shown in Fig. 8. For small velocities (u = 1 m/s) can be observed small values of the heat transfer coefficient, which can explained to be near to natural convection heat transfer.

The overall heat transfer coefficient dependence in function of inlet air flow velocity get in m/s, and which can be expressed by the relation:

$$h_{overall} = 4.857 + 4.143 \cdot u_{in} - 0.0357 \cdot u_{in}^2 \text{, W/m}^2\text{.K} \quad (8)$$

4. Conclusions

1. The use of combined conduction, forced convection and the turbulence in solid-fluid thermal interaction proved a method to evaluate the engine surface temperatures and heat transfer coefficient for air flowing over external engine surface.
2. The air velocity significantly influenced the heat transfer by showing the change from natural convection to forced turbulent convection in external flow.
3. A regression relation for overall heat transfer coefficient in function of inlet air velocity is obtained.

REFERENCES

* * * COMSOL Multiphysics ver. 3.5a, 2008.
Abedin M.J., Masjuki H.H., Kalam M.A., Sanjid A., Ashrafur Rahman S.M., Masum B.M., *Energy Balance of Internal Combustion Engines Using Alternative Fuels*. Renew. Sust. Energ. Rev., **26**, 20−33, 2013.
Heywood J.B., *Internal Combustion Engine Fundamentals*. Mc-Graw Hill, New York, 1988.
Martyr A.J., Plint M.A., *Engine Testing: Theory and Practice*. Butterworth Heinmann, London, 2007.
Sandu V., Ungureanu V.B., Ivanoiu M., Costiuc L., *Experimental Analysis on External Surface Convective-Radiative Heat Transfer in Stationary Diesel Engines*. Rev. Termotehnica, **1**, 32−36, 2014.

SIMULAREA NUMERICĂ A TRANSFERULUI DE CĂLDURĂ PE SUPRAFAŢA EXTERIOARĂ A UNUI MOTOR DIESEL STAŢIONAR

(Rezumat)

Această lucrare urmăreşte simularea numerică a transferului de căldură pentru obţinerea unei relaţii de calcul a fluxului termic de convectiv cedat mediului exterior prin suprafaţa exterioară a unui motor Diesel staţionar (de laborator). Simularea a fost realizată în mediul de programare CFD numit COMSOL Multiphysics. Pentru simulare au fost considerate condiţii externe de curgere pentru convecţie naturală şi convecţie forţată datorate ventilatorului motorului. Câmpul de temperatură al suprafeţei exterioare a motorului a fost măsurat cu ajutorul termografiei cu infraroşu şi a fost folosit ca valoare iniţială pentru modelul simulat. Modelul a luat în considerare curgerea turbulentă prin folosirea modelului k-eps de turbulenţă. Au fost reţinute din rezultatele obţinute evoluţia câmpului de temperatură pentru u = 10 m/s pentru a evidenţia apariţia influenţei regimului turbulent, criteriile de similitudine Reynolds şi Nusselt reprezentate în funcţie de viteza de intrare a aerului şi în lungul domeniului de perete solid. De asemenea, s-a reprezentat variaţia coeficientului local şi global de transfer de căldură în funcţie de viteza aerului, precum şi o relaţie de calcul obţinută prin regresie numerică.