Multifidelity simulation of underhood thermal system for a bus engine


Published in:
Applied Thermal Engineering

Document Version:
Peer reviewed version

Queen's University Belfast - Research Portal:
Link to publication record in Queen's University Belfast Research Portal

Publisher rights
Copyright 2021 Elsevier. This manuscript is distributed under a Creative Commons Attribution-NonCommercial-NoDerivs License (https://creativecommons.org/licenses/by-nc-nd/4.0/), which permits distribution and reproduction for non-commercial purposes, provided the author and source are cited.

General rights
Copyright for the publications made accessible via the Queen's University Belfast Research Portal is retained by the author(s) and / or other copyright owners and it is a condition of accessing these publications that users recognise and abide by the legal requirements associated with these rights.

Take down policy
The Research Portal is Queen's institutional repository that provides access to Queen's research output. Every effort has been made to ensure that content in the Research Portal does not infringe any person's rights, or applicable UK laws. If you discover content in the Research Portal that you believe breaches copyright or violates any law, please contact openaccess@qub.ac.uk.

Open Access
This research has been made openly available by Queen's academics and its Open Research team. We would love to hear how access to this research benefits you. – Share your feedback with us: http://go.qub.ac.uk/oa-feedback
Multifidelity Simulation of Underhood Thermal System for a Bus Engine

Konstantinos Karamanos\textsuperscript{a,b}, Ehsan Afrasiabian\textsuperscript{a,b}, Sung In Kim\textsuperscript{a,*}, Roy Douglas\textsuperscript{a,b}, Yasser Mahmoudi\textsuperscript{a,c,**}

\textsuperscript{a}School of Mechanical and Aerospace Engineering, Queen’s University Belfast, BT9 5AH, United Kingdom
\textsuperscript{b}Bamford Technology Centre (W-Tech), Queen’s University Belfast, BT9 5BS, United Kingdom
\textsuperscript{c}Department of Mechanical, Aerospace and Civil Engineering, The University of Manchester, M13 9PL, United Kingdom

Abstract

This study performs a combined 0-dimensional/3-dimensional modelling approach to investigate the fluid flow and heat transfer characteristics of bus thermal management systems. The 3-dimensional model is deployed to develop new correlations for the heat transfer coefficient (Colburn-j factor) and the friction factor (Fanning-f factor) at the air-side of the multi-louver radiator and charge-air cooler. The effect of the fan operation is also taken into account. The existing correlations in the literature developed for cars where the radiator and charge-air cooler are placed in the front section of the vehicle exposed to a uniform incoming air flow. While in buses, these components are placed at the vehicle rear section and in contact with a turbulent and non-uniform air flow, highlighting the need for development of new Colburn-j factor and Fanning-f factor for air flow within the louvered fins in these two components. The coefficients developed are incorporated into the 0-dimensional model to predict the thermal characteristics of the bus underhood for a range of operating conditions. The 0-dimensional model simulates the heat interaction of the multiple thermodynamic systems. Thus, a better understanding of the thermal management is achieved by investigating the

\textsuperscript{*}Corresponding author
\textsuperscript{**}Corresponding author

Email addresses: s.kim@qub.ac.uk (Sung In Kim), yasser.mahmoudi@manchester.ac.uk (Yasser Mahmoudi)
energy distribution within the engine compartment and describing the performance of the thermal systems. The 0-dimensional/3-dimensional model is examined under the peak brake power condition. A coolant mass flow rate of 3.74 kg/s and fans speed of 4000rpm are the most optimum results since the coolant’s temperature is decreased by 5°C and the parasitic losses are kept at minimum.

*Keywords:* Thermal management, 0D/3D simulations, Heat transfer coefficient, Friction factor

---

**Nomenclature**

**Latin Symbols**

- $A$: Area ($m^2$)
- $C_f$: inertial coefficient
- $C_d$: coefficient of drag
- $C_p$: specific heat at constant pressure ($J/kgK$)
- $C_r$: heat capacity ratio
- $D$: diameter ($m$)
- $f$: Fanning friction factor
- $F_p$: fin pitch ($m$)
- $F_t$: fin thickness ($m$)
- $h$: heat transfer coefficient ($W/m^2K$)
- $I_c$: Turbulence Intensity
- $j$: Colburn factor
- $k$: thermal conductivity ($W/mK$)
- $k_{f,eff}$: effective thermal conductivity of fluid part in porous media ($W/mK$)
$k_{s, eff}$ effective thermal conductivity of solid part in porous media (W/mK)

$l$ length (m)

$m$ mass flow rate (kg/s)

$N$ Engine speed (rev/s)

$Nu$ Nusselt number

$P$ pressure (Pa)

$Pr$ Prandl number

$Q$ heat transfer rate (W)

$Re$ Reynolds number

$St$ Stanton number

$T$ temperature (K)

$u$ velocity (m/s)

$W$ power (W)

**Greek Symbols**

$\alpha$ specific surface area (m$^{-1}$)

$\Delta P$ pressure drop (Pa)

$\epsilon$ effectiveness

$\eta$ efficiency

$\gamma$ isentropic exponent

$\kappa$ permeability (m$^2$)

$\mu$ viscosity (kg/ms)

$\pi$ pressure ratio

$\rho$ density (kg/m$^3$)
\[ \tau \] engine torque (Nm)
\[ \tau_c \] cooling coefficient

**Subscripts**

- \( amb \) ambient
- \( lp \) brake power
- \( c \) cross section
- \( comp \) compressor
- \( cyl \) cylinder
- \( eff \) effective
- \( em \) exhaust manifold
- \( exh \) exhaust
- \( f \) fluid
- \( fs \) fluid - solid
- \( h \) hydraulic
- \( im \) intake manifold
- \( LHV \) lower heating value
- \( lp \) louver pitch
- \( m \) mean
- \( min \) minimum
- \( rad \) radiator
- \( s \) solid
- \( sf \) surface
- \( w \) wall
1. Introduction

The automotive industry encounters new challenges of enhancing fuel economy and decreasing emissions, in order to meet the environmental targets. Engine’s thermal management has a vital role in tackling these issues. The thermal management systems in diesel engines consist of the engine cooling sub-system, the turbocharged air cooling sub-system, the engine lubrication sub-system and the intake and exhaust sub-system including the Exhaust Gas Recirculation (EGR) cooling [1]. Two of the main components in the cooling pack, the radiator and charge-air cooler, enhance engine performance and prevent mechanical failure [2]. These components consist of an axial fan and multi-louver fined tube heat exchangers.

Due to the significance of these two components in the engine’s thermal management, previous research has been dedicated to better understand the flow thermal features of the louver fins and develop correlations for the heat transfer coefficient and friction factor. Davenport [3], Achaicha and Cowell [4] and Sunden and Svantesson [5] analysed experimentally the heat and airflow features of several louvered fin heat exchangers and developed empirical correlations for heat transfer and pressure drop in the form of Colburn-j and Fanning-f factor respectively. Webb and Jung [6] reported that the thermal performance of a louvered fin is 50% higher than a conventional plate tube. Rugh et al. [7] investigated the heat transfer and pressure drop characteristics of high density louvered fin and found out that there is 25% and 110% increase respectively compared to a plain fin. The most accurate empirical correlations for heat transfer and pressure drop have been developed by Kim and Bullard [8], Chang and Wang [9] and Chang et al. [10].
conducted experiments on 45 different heat exchangers while Chang and Wang developed generalised correlations based on a data bank consisting of 91 samples. While Li and Wang developed correlations of heat transfer coefficient and pressure drop for multi-region louver fins.

These correlations have been extensively used by previous studies that investigated the heat transfer at a fined tube heat exchanger by modelling it as porous media. Hur et al. instead of using the empirical correlations, proposed another method which includes the tubes geometry and solves conjugate heat transfer. Although this method generated accurate results, it depended on experimental data for heat transfer and friction coefficients. To eliminate the need of experimental correlations Lee et al. suggested a multi-scale method which initially investigates the heat and fluid flow in microscopic level and the generated data are transferred to the porous media. However, as the heat exchangers are normally placed at the downstream of the fans, the fan operation has significant effect on the inlet boundary conditions of the microscopic model.

A full-scale 3D modelling using computational fluid dynamics (CFD) approach for vehicle’s thermal systems is complex and time consuming. Thus an integrated 0D/3D modelling technique is suggested. The components that are exposed to the external air in the cooling pack, including radiator, charge-air cooler and evaporator, are modelled in 3D while the rest of the systems in 0D. Previous studies have recommended a co-simulation where the 0D model transfers data to 3D model and vice versa. Other studies carried out initially the CFD simulations and generate maps of data and feed them to the 0D model to complete the thermal analysis.

In literature, the majority of research has been conducted on cars or trucks where radiator and charge-air cooler are placed in the front section of the vehicle. Hence, the inlet airflow is relatively uniform, laminar and perpendicular to the inlet face. Whereas, these two components in a bus are located in the rear part which are exposed to a highly non-uniform and turbulent external flow which enters the engine cabin by the side of the vehicle. Therefore the aim of this study was to develop of a numerical tool that investigates the heat transfer characteristics and fluid flow features of a bus engine. 3D CFD simulations were conducted to estimate the heat transfer coefficient and friction factor of the radiator’s and charge-air cooler’s air side and to develop new correlations for the Colburn-j and Fanning-f factors. These correlations then were fed to the 0D model to complete the
thermal analysis. The CFD software used for the 3D simulations is ANSYS FLUENT 19.2 while the 0D model is developed using MATLAB. After the CFD simulations new correlations for Colburn-$j$ and Fanning-$f$ factors are developed and fed to the 0D model to complete the thermal calculation.

2. Model Description

The analysis of this study mainly focuses on the heat and fluid flow in the underhood thermal management systems in a diesel bus engine. The 0D MATLAB based model consists of the different engine sub-systems, including the engine cooling, turbocharger cooling, engine lubrication and the intake and exhaust sub-systems including the EGR cooling. Thermodynamic laws and heat transfer aspects are considered, in order to analyse the energy distribution in the bus underhood system.

![Figure 1: Schematic diagram of the bus thermal management](image)

The cooling pack components, including the radiator, the charge-air cooler and the axial fans are located at the rear side of the vehicle and they are exposed to a turbulent and transient airflow. Therefore, high fidelity 3D simulations are needed in order to predict the airflow and temperature distribution across these two components, and consequently develop correlations.
for the local heat transfer coefficient and friction factor in the louvered fins within these two components. Data are transferred from the 3D CFD simulation to 0D model via the newly developed correlations. Figure 1 shows a diagram of the engine components and utilization of the 0D/3D model. The algorithm of the 0D/3D model is illustrated in Figure 2 while further details about the model can be found in the Appendix A.

Figure 2: Algorithm of the 0D/3D model

2.1. CFD Simulations

Considering the complicated and multi-scale nature of the bus thermal systems, three stage CFD models are utilised. Figure 3 illustrates the three CFD models and the data required for each one. The first one analyses the airflow around the bus and estimates the air properties and turbulent parameters at the grilles which is the inlet of the engine compartment. These data are fed to the second CFD model which investigates the flow within the cooling pack to investigate the flow behaviour between the fans and the heat exchangers (i.e. radiator and charge-air cooler).

Similarly to the first CFD model, the second model calculates the air properties and turbulent parameters at the downstream of the fans i.e. upstream of the radiator and charge-air-cooler. Finally, the third model utilises
the outputs of the second model and solves the flow and heat transfer characteristics in a louver fin in the radiator and charge-air cooler. The Reynolds-averaged Navier-Stokes equations are solved for all the simulations. The flow was at steady state conditions and it was modelled as incompressible and viscous.

continuity equation:
\[
\frac{\partial (\rho u_i)}{\partial x_i} = 0
\]  

momentum equation:
\[
\frac{\partial (\rho u_i u_j)}{\partial x_i} = \frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_i}{\partial x_i} \right) \right) + \frac{\partial}{\partial x_j} \left( -\rho u_i u'_j \right)
\]

energy equation:
\[
\frac{\partial (\rho u_i T)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \frac{k}{C_p} \frac{\partial T}{\partial x_j} \right) + S_T
\]

The full implicit method for pressure-based equations (Coupled) algorithm was used for all the three models. The convective terms of the momentum, turbulence model and energy equations are discretized using a second order upwind (SOU) scheme, while the diffusion terms are central-differenced.

2.1.1. Bus Model (CFD #1)

As the engine bay of the bus is located at the rear end and the air enters from the side a high fidelity model is necessary to capture the flow features. All simulations were run at 10 km/h bus speed, which is the most frequent speed in the UK cities [22]. Figure 4 illustrates a representation of the computational domain highlighting its dimensions. The whole bus is located in the centre of the domain replicating a wind tunnel test. The cross sectional area blockage ratio is 3.6%. The boundary conditions are velocity inlet, pressure outlet at atmospheric pressure, no-slip moving wall for the bottom side,
The turbulence model is the k-ω SST and the mesh is unstructured with prism layers (shown in Fig. 5) in order to capture the boundary layers properly. Table 1 shows a grid independence study and the model’s validation against literature for the bus drag coefficient [23]. The axial velocity examined for the results is perpendicular to grilles. Grid resolution of 16 million elements was chosen for the simulations as a higher number of elements did not produce reasonably different results.

![Computational domain of CFD #1 (bus model)](image)

Figure 4: Computational domain of CFD #1 (bus model), where L, h and w are the length, height and width of the bus respectively (not to scale)

![CFD #1 Details of the mesh](image)

Figure 5: CFD #1 Details of the mesh (a) finer mesh to capture the flow features close to the bus and the wake (b) boundary layer

### 2.1.2. Cooling Pack Model (CFD #2)

The next CFD simulations considered the cooling pack of the bus, i.e. axial fans, radiator and charge-air cooler. The boundary conditions are pressure inlet, ambient pressure outlet and slip walls (Fig. 6). The axial fans
Table 1: CFD #1 Grid Sensitivity Analysis and Validation

<table>
<thead>
<tr>
<th>Number of elements</th>
<th>Axial velocity at grilles (m/s)</th>
<th>$C_d$</th>
<th>$C_d$ Experimental [23]</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.5 million</td>
<td>0.456</td>
<td>0.500</td>
<td></td>
</tr>
<tr>
<td>9.9 million</td>
<td>0.467</td>
<td>0.467</td>
<td></td>
</tr>
<tr>
<td>16.1 million</td>
<td>0.473</td>
<td>0.463</td>
<td></td>
</tr>
<tr>
<td>21.2 million</td>
<td>0.474</td>
<td>0.461</td>
<td></td>
</tr>
</tbody>
</table>

are modelled using the Multiple Reference Frame (MRF) method. In this method the fan is placed in a specified rotating domain. Inside the rotating domain the flow is modelled using an accelerating frame while outside of it using a non-accelerating frame. This is an accurate steady state approach with low computational cost. The heat exchangers are modelled as porous media, with the flow forced to move in a direction perpendicular to the inlet face. The porous coefficients were provided by the louver fin model. More details are included at section 2.1.3 The turbulence model used is the realisable k-\epsilon model with enhanced wall treatment.

Table 2: Geometric parameters of the radiator, charge-air cooler and fans

<table>
<thead>
<tr>
<th>Radiator</th>
<th>Charge-air cooler</th>
</tr>
</thead>
<tbody>
<tr>
<td>Height</td>
<td>712 mm</td>
</tr>
<tr>
<td>Width</td>
<td>445 mm</td>
</tr>
<tr>
<td>Depth</td>
<td>450 mm</td>
</tr>
<tr>
<td>No. of tubes</td>
<td>28</td>
</tr>
<tr>
<td>Diameter</td>
<td>220 mm</td>
</tr>
<tr>
<td>Hub - Tip ratio</td>
<td>0.4</td>
</tr>
<tr>
<td>No. of fans</td>
<td>4</td>
</tr>
</tbody>
</table>

The heat exchangers consist of the tubes filled with the working fluids, tubes walls and the porous media zone which substitutes the complex fin geometry. The porous media method has been proven to be accurate with low computational cost [18]. The governing equation (4) of the porous media coming from the modified Darcy equation by Brinkman-Forchheimer, where the first part considers the viscous effect while the second part the inertia effect:
\[ \nabla P = \frac{\mu_f}{\kappa} \nabla^2 u - \frac{\mu_f}{K} u - \frac{\rho_f \kappa C_f}{\sqrt{K}} |u|u \] (4)

The viscous and inertia coefficients required for the porous media model are calculated using the data provided from the louver fin model in the section 2.1.3. To calculate the temperature field in the porous media the Local Thermal Non-Equilibrium (LTNE) method was used through solving two separate energy equations for the fluid and solid phases as:

Fluid zone:

\[ (\rho C_p)_f u \nabla T_f = \kappa \nabla (k_{f,\text{eff}} \nabla T_f) + h_{fs} a_{fs} (T_s - T_f) \] (5)

Solid zone:

\[ 0 = (1 - \kappa) \nabla (k_{s,\text{eff}} \nabla T_s) + h_{fs} a_{fs} (T_f - T_s) \] (6)

where \( h_{fs} \) is the internal heat transfer coefficient between the fluid and solid phases and \( a_{fs} \) is the specific surface area (surface area per unit volume). The effective thermal conductivities of fluid \( (k_{f,\text{eff}}) \) and solid \( (k_{s,\text{eff}}) \) are calculated using the equations proposed by Song et al. [15].

\[ k_{f,\text{eff}} = \kappa k_f = \frac{1 - F_t}{F_p} k_f \] (7)

\[ k_{s,\text{eff}} = (1 - \kappa) k_s = \frac{F_t}{F_p} k_s \] (8)

The fluid domain (see Fig. 6) close to the fans was meshed with an unstructured mesh due to fan’s geometrical complexity while the rest of the fluid domain, including the porous media, was modelled with a structured mesh. Initially the axial fan and the porous media were modelled separately for validation purposes. The axial fan was validated in respect to the velocity distribution at the downstream. Figure 7 shows a comparison between the CFD results, with different meshes, produced in this work and the experimental data and CFD simulations conducted by Franzke and Sebben [24]. The results show a small discrepancy due to the slightly different blades profile used. Details about the validation of the porous media can be found in Figure 14 in chapter 3.2. The final grid, including fluid domain, the heat exchangers as porous media and all the fans, consists of approximately 18 million elements.
Figure 6: CFD #2 (a) Computational domain of cooling pack including fans, charge air cooler and radiators (modelled as porous media) (b) mesh downstream of the fans.

Figure 7: Axial Velocity (perpendicular to fans) at 5 m/s, 1400rpm, 4cm at the downstream of the fan (a) centreline (b) 10 cm to the right.

2.1.3. Louver Fin Model (CFD #3)

As mentioned in the section 2.1.2, the heat transfer coefficient and pressure drop are obtained by the louver fin model which takes into account the actual fin geometry (see Fig. 8).

Table 3: Geometric parameters of the louver fin

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Louver angle</td>
<td>28 degree</td>
</tr>
<tr>
<td>Louver length</td>
<td>13 mm</td>
</tr>
<tr>
<td>Fin pitch</td>
<td>2.1 mm</td>
</tr>
<tr>
<td>Fin depth</td>
<td>71.5 mm</td>
</tr>
<tr>
<td>Tube pitch</td>
<td>26.8 mm</td>
</tr>
<tr>
<td>Louver pitch</td>
<td>3 mm</td>
</tr>
<tr>
<td>Fin thickness</td>
<td>0.16 mm</td>
</tr>
<tr>
<td>Fin length</td>
<td>19.6 mm</td>
</tr>
<tr>
<td>Hydraulic diameter</td>
<td>3.04 mm</td>
</tr>
</tbody>
</table>
The boundary conditions are velocity inlet, pressure outlet, symmetry at the sides and periodic for bottom and top (see Fig. 13). The tubes wall temperature is fixed at 358K, as it does not affect the flow features [18]. The velocity inlet and the turbulence parameters (i.e. turbulence intensity and turbulent viscosity ratio) are obtained by the cooling pack model (CFD #2). Grid sensitivity analysis of the louver fin model (Table 4) is carried out and it was concluded that a mesh size greater than 6.2 million (shown in Fig. 8b) does not affect the results.

![Image](a) CFD #3 Louver fin geometry including the tube’s wall (b) mesh at the vertical plane

Figure 8: (a) CFD #3 Louver fin geometry including the tube’s wall (b) mesh at the vertical plane

<table>
<thead>
<tr>
<th>Number of elements</th>
<th>Colburn-j factor</th>
<th>Fanning-f factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>5 million</td>
<td>0.0110</td>
<td>0.0616</td>
</tr>
<tr>
<td>6.2 million</td>
<td>0.0098</td>
<td>0.0557</td>
</tr>
<tr>
<td>10.3 million</td>
<td>0.0097</td>
<td>0.0555</td>
</tr>
</tbody>
</table>

Table 4: CFD #3 Grid Sensitivity Analysis at Re = 1778.5

Non dimensionalised parameters have been utilised to calculate the heat transfer rate and pressure drop in the fins system. In particular, the Colburn-j factor and Fanning-f factors have been used as a measure for the heat transfer coefficient and friction factor respectively, using the following equations:

\[ Q = hA(T_w - T_m) \]  
\[ T_m = \frac{(T_w - T_{in}) - (T_w - T_{out})}{\ln\left(\frac{T_w + T_{in}}{T_w - T_{out}}\right)} \]

The Colburn-j factor is then calculated using the following equation:
\[ j = StPr^{2/3} = \frac{h}{\rho u_{\text{max}} C_p} \left( \frac{C_p \mu}{k} \right)^{2/3} \]  

The Fanning-f factor is defined as:

\[ f = \frac{2\Delta P}{\rho u_{\text{max}}^2 A_c} \frac{A_c}{A_{sf}} \]  

Simulations were performed for seven cases with different velocities (range from 2 to 14 m/s). Then correlations were developed for the Colburn-j and Fanning-f factors. These correlations are then utilised in the 0D model which deployed to analyse the bus thermal management systems.

2.2. 0D Model

The 0D MATLAB based model consists of five sub-models: the engine model, the engine cooling model, the turbocharged cooling model, the lubricant model and the exhaust model. The sub-models are integrated to each other and exchange heat flows in order to calculate the energy distribution within the thermal systems. The properties of the working fluids are obtained by REFPROP \[25\]. The inputs of the model are the driving and environmental conditions, i.e. engine speed, engine torque, ambient temperature and pressure.

2.2.1. Engine Energy Balance

To understand how the energy supplied to the engine is distributed among different components in the engine bay, a detailed energy balance analysis is carried out. Figure 9 shows how the fuel energy is converted to useful brake power, mechanical losses, heat rejection to coolant and exhaust gas heat. Part of the mechanical losses, such as friction between the pistons and cylinder's wall, is reduced by the lubricant while the rest is lost as heat in the engine bay \[20\]. Initially, an amount of heat carried from the coolant is transferred to the lubricant in order to cool it down, while the rest of it is exchanged with the ram air at the radiator. A fraction of the exhaust gas is recovered through the EGR valve and heat is transferred between the exhaust gas and the coolant via the EGR cooler. The rest of the exhaust gas rotates the turbine at the turbocharger before it is lost in the environment. The turbine then drives the compressor recovering in this way some part of the waste energy. As the compressed air needs to cool down before it is supplied
to the engine, heat is lost at the charge-air cooler. Normally the resultant exhaust gas from the turbine is lead to the after-treatment system to reduce the emissions, however in this study the effect of the after-treatment system is not considered [27].

\[ Q_{fl} = W_{bp} + W_{mec} + Q_{rej} + Q_{exh}, \]  
\[ Q_{fl} = \dot{m}_{fl} Q_{LHV}, \]

\[ Q_{fl} = W_{bp} + W_{mec} + Q_{rej} + Q_{exh}, \]  
\[ Q_{fl} = \dot{m}_{fl} Q_{LHV}, \]
\[ W_{bp} = 2\pi N\tau. \]  

(15)

The friction losses are assumed to be a polynomial function of engine speed, where the tuning parameters are determined by solving a non-linear least equations problem [29]. The parasitic losses are estimated from the pressure rise across the pumps and fans by the equation 16, where the efficiency of pump and fan is 0.75 and 0.8 respectively [30, 31].

\[ W_{pump/fan} = \frac{m_f \Delta P}{\rho \eta}. \]  

(16)

The exhaust gas heat is calculated by the enthalpy difference between the exhaust and the intake. The difference of composition between the exhaust gas and atmospheric air is not taken into account and the working fluid is considered as ideal gas [28, 32].

\[ Q_{exh} = ((\dot{m}_f + \dot{m}_a)C_pT)_{em} + (\dot{m}_aC_pT)_{im}, \]  

(17)

The amount of heat rejected from the combustion chamber to cylinder’s wall is estimated by

\[ Q_{rej} = hA(T_{cyl} - T_c), \]  

(18)

where \( h \) is the convective heat transfer coefficient, \( A \) is the cylinder’s area, \( T_{cyl} \) and \( T_c \) is the surface temperature of cylinder’s wall and the coolant temperature respectively. To estimate the heat transfer coefficient using the empirical correlation developed by Hohenberg [33], a quasi-static engine model which considers a thermodynamic analysis at different crank angles has been developed.

\[ h_g = C_1 U^{-0.06} P^{0.8} T^{-0.4}(C_2 + S_p)^{0.8} \]  

(19)

where \( S_p \) is the piston velocity and constants \( C_1 = 130 \) and \( C_2 = 1.4 \). The heat transfer from the walls to the coolant is mainly due to convection [34].

\[ h_c = 0.023Re^{0.8} Pr^{0.4} \frac{k}{D} \]  

(20)

Figure 9 illustrates that the fuel energy input is basically converted to work output and heat losses. Part of the exhaust gas is utilised by the turbocharger which leads to some heat losses at the charge-air cooler. Similarly
the heat rejected from the engine is divided to coolant and oil. There are also heat losses in the engine bay from the engine, turbocharger and cooling system. A new equation can be written (equation 21).

\[ Q_s = W_{bp} + Q_{rad} + Q_{oil} + Q_{exh} + Q_{cac} + Q_{egr} + Q_{misc}, \] (21)

where \( Q_{rad} \) and \( Q_{oil} \) is the heat transfer rate at radiator and oil cooler respectively, \( Q_{cac} \) is the heat transfer at the charge-air cooler, \( Q_{egr} \) is the heat transfer at EGR cooler and \( Q_{misc} \) is the miscellaneous heat losses.

### 2.2.3. Compressor

Given the environmental and operating conditions the mass flow rate of the charged air is calculated via a compressor map provided by the manufacturer while its temperature using the equation 22, assuming compressor’s isentropic efficiency of 0.7 and adiabatic compression process.

\[ T_{out} = T_{amb} \left( 1 + \frac{\pi}{\gamma} - 1 \right) \] (22)

where \( \pi \) is the pressure ratio, \( \gamma \) the isentropic exponent and \( \tau_c \) the cooling coefficient of external cooling [14]. The pressure ratio is calculated by the compressor map.

### 2.2.4. Radiator and Charge-air Cooler

Both radiator and charge-air cooler are modeled to be finned tube heat exchangers. For the air side the correlations developed by the CFD simulations in the section 3.3 are utilised for heat transfer and friction coefficient. For the tube side which includes the coolant and charged-air the heat transfer coefficient is predicted in the form of Nusselt number using the empirical correlation developed by Gnielinski [35].

\[ Nu = \frac{\frac{f}{2}(Re - 1000)Pr}{1 + 12.7\sqrt{\frac{f}{5}(Pr^{2/3} - 1)} \left( 1 + \left( \frac{D_h}{l} \right)^{2/3} \right)} \] (23)

The overall heat transfer coefficient is calculated using the thermal resistance approach. Thus, the heat rejection at both heat exchangers is predicted using the effectiveness-number of transfer unit (NTU) approach considering them as counter-flow heat exchangers [36].
\[ Q = (mC_p)_{\text{min}} \epsilon (T_f - T_{\text{air}}) \] (24)

where \( m \) and \( C_p \) are the mass flow and specific heat of the fluid with the lowest values, \( T_f \) and \( T_{\text{air}} \) are the temperatures of the working fluid and air respectively, and \( \epsilon \) is the effectiveness of the heat exchanger which is estimated by equation:

\[ \epsilon = 1 - e^{\frac{1}{0.22} e^{(-C_T NTU)^{0.78}} - 1} \] (25)

The developed 0D model is used to investigate the thermal energy distribution in the bus engine with the aim of reducing the energy losses. By analysing the thermal systems, failure of components under extreme conditions can be avoided as well as a better fuel economy can be achieved.

3. Results

3.1. CFD Results

Figure 10 shows the contour of air velocity near the grilles obtained from the CFD simulation for the bus speed of 10 km/h (2.77 m/s). Figure 10 shows a highly non-uniform distribution of air velocity, varies from the initial air velocity equal to 2.6 m/s at the top region of the grilles to 0.1 m/s near the bottom region.

![Simulated velocity at the grilles and around the bus at 10km/h](Figure 10)

Figure 10: Simulated velocity at the grilles and around the bus at 10km/h

Figure 11 shows the velocity distribution downstream of the fans. Due to the geometry of the engine block, there is a high velocity at the bottom of the heat exchangers (radiator and charge-air cooler) while a reversed flow also occurs. Area averaged values were transferred to the louver fin model to continue the simulations.
3.2. Validation of Louver Fin and Porous Media

For validation purposes the louver fin CFD model was tested against the empirical correlations developed by Kim and Bullard [8], Chang and Wang [9] and Chang et al. [10]. Figure 12 shows that the CFD results are in good agreement with these empirical correlations.

The highest error is at low Reynolds numbers ($Re_{lp} < 1200$), where the flow is considered laminar [37], due to the simulation with turbulent modeling. However, these low Reynolds numbers rarely occur at buses due to the heavy operation of the fans. The maximum relative error for the Fanning-$f$ factor is 57% and 17% against [8] and [10] respectively ($Re_{lp} = 444$). While
the higher the Reynolds number the minimum the error (2.3% and 0.2% at $Re_{lp} = 3112$). Similarly the maximum error for the Colburn-j factor occurs at $Re_{lp} = 444$ (19% and 17% against [9] and [8] respectively). Whereas the relative error for the rest of Reynolds numbers fluctuates between 0.03% and 8%, with the minimum one at $Re_{lp} = 1339$ for both empirical correlations.

For the validation of the porous media model, the pressure drop and temperature difference in the fins region obtained from the porous media model are compared against those from the detailed CFD of the louvered fin model.

Figure 13: Computational domain and boundary conditions of (a) louver fin (b) porous media

Figure 14: Difference across the porous media (a) Pressure drop (b) Temperature difference

As shown in Figure ??, the computational domain and boundary conditions were identical. However, the geometry of the louver fin was substituted by a porous zone. This resulted in 86.6% decrease in mesh size. The results between the two models (Fig. 14) show a good agreement with the maximum error 15% and 13.5% for pressure drop and temperature difference respectively. Once the porous zone was validated, the viscous and inertial
coefficients were obtained and fed to the cooling pack model (CFD #2) in order to acquire the correct airflow features downstream of the fan. Subsequently, the airflow properties were transferred back to the louver fin model to continue the analysis.

3.3. New Correlations of Colburn-$j$ and Fanning-$f$ Factors

Since both models of the axial fan and the heat exchangers (porous media) were validated, they were combined in order to carry on the analysis. The boundary conditions were established by the CFD bus model at bus speed 10km/h. The fans operated at different speeds and an average value for each speed was fed back to the louver fin model. With this approach the effect of turbulence, created by the fans, on heat transfer and friction coefficient is taken into account. Figure 15 shows the temperature and pressure contours at different velocities. A higher velocity in the middle of the louver fin occurs at higher turbulence intensity. Moreover, higher pressure at the first louvers is generated.

![Figure 15: Contours of velocity, pressure and temperature at (a) Re = 1289, $I_c = 13.4\%$ (b) Re = 1289, $I_c = 0.1\%$ (c) Re = 2290, $I_c = 17.8\%$ (d) Re = 2290, $I_c = 0.1\%$](image)

The results in Figure 16 show that at low fan speed turbulence was not generated. Thus, the fan operation does not have a major effect in either Colburn-$j$ or Fanning-$f$ factor. However, at higher Reynolds number the perturbation created by the fan has a significant effect which leads to higher values. Therefore new correlations for the Colburn-$j$ (26) and Fanning-$f$ (27)
Figure 16: Validation of the louver fin model against empirical correlations (a) Fanning $f$-factor (b) Colburn $j$-factor

Factors are proposed for the present louver fin as a function of Reynolds number. The coefficients were obtained using the least squares method, where the coefficient of determination is 0.993 and 0.991 for Colburn-$j$ and Fanning-$f$ factor respectively.

$$j = 0.2414Re_{lp}^{-0.4193}$$  \hspace{1cm} (26)  

$$f = 0.517Re_{lp}^{-0.2897}$$  \hspace{1cm} (27)

3.4. 0D/3D Results

Feeding the 0D model with the new correlations for heat transfer coefficient and friction factor Eqs. 26 and 27, respectively, the thermal energy balance is carried out for the diesel engine. Flow distribution in the underhood affects radiator’s and charge-air cooler’s thermal performance. Moreover, due to the location of the engine compartment, the fan plays a vital role in regulating the flow distribution. Especially when the heat load is high or when the bus speed is low.

Figure 17 shows the effect of coolant mass flow rate and fan speed at the coolant temperature difference under the condition of maximum brake power (155kW). Seven fan speeds ranging from 1000 to 7000rpm were tested, while the coolant pump’s speed range was from 1000 to 3000rpm. Increasing the mass flow rate can assist transferring higher amount of heat up until a point where its effect becomes insignificant. However, an increase in parasitic losses also occurs and its effect must be taken into account (Fig. 17). Hence, a coolant mass flow rate of 3.74 kg/s at a fan speed of 4000rpm is recommended for the engine’s cooling. The coolant temperature is decreased by 5°C which
is sufficient to keep the engine operating at a proper temperature range as well as avoiding a thermal shock of the radiator. Due to pump’s lower power consumption, it is preferred to increase the pump speed instead of the fan speed in order to achieve the same temperature drop. This will result in higher brake power output.

Figure 17: (a) Temperature difference of the radiator’s coolant side for different pump speeds: 1000rpm = 2.88kg/s; 2000rpm = 3.74kg/s; 3000rpm = 4.48kg/s (b) Parasitic losses at maximum brake power for different pump speeds: 1000rpm = 2.88kg/s; 2000rpm = 3.74kg/s; 3000rpm = 4.48kg/s
4. Conclusion

In the present work, the heat and fluid flow of the underhood thermal systems of a bus engine is investigated using an integrated 0-dimensional/3-dimensional model. The heat transfer coefficient and friction factor of the air side of radiator and charge-air cooler are estimated with the aid of 3-dimensional simulations. In this way the turbulent flow phenomena are captured whilst the effect of fans is taken into consideration.

The 3-dimensional model is deployed to develop new correlations for the heat transfer coefficient (Colburn-j factor) and the friction factor (Fanning-f factor) at the air-side of the multi-louver radiator and charge-air cooler. Initially, a 3-dimensional bus model was developed which considers the bus geometry and captures the flow entering the engine cabin. These flow features are transferred to the cooling pack model which takes into consideration the effect of the cooling fans and analyses the flow downstream of the fans, i.e. upstream of the radiator and charge-air-cooler. Finally, these data are used as inputs to the fin model which estimates the heat transfer coefficient and friction factor. The newly developed correlations are fed to the 0-dimensional model which simulates the heat interaction of multiple components in the bus underhood.

Through this approach pumps and fans speeds are adjusted to investigate the effect of the working fluids mass flow rate and achieve a better thermal management at different vehicle operating conditions. A coolant mass flow rate of 3.74 kg/s and fans speed of 4000rpm are the most optimum results in the tested case since the coolant’s temperature is decreased by 5°C and the parasitic losses are kept at minimum.

5. Acknowledgements

This work was supported by the Engineering & Physical Sciences Research Council (EPSRC) and Bamford Technology Centre (W-Tech) at Queen’s University Belfast.

References


[34] F. Dittus, L. Boelter, Heat transfer in automobile radiators of the tubular type, Univ Calif Publ Eng 2 (1930) 443–461.


Appendix A  Modelling Procedure

Figure A.1 shows the modelling procedure used in CFD and MATLAB environment.

Figure A.1: Flow chart of the multi-fidelity model