KES Transactions on Sustainable Design and Manufacturing I
Sustainable Design and Manufacturing 2014 : pp.79-82 : Paper sdm14-046

Numerical Modelling of a Marine Vessel Engine Room with Field Measurements

Will Newton[1], Mel Lewis[2], Nick Lavery, Ben Evans, David Bould, Johann Sienz

[1]ASTUTE, Digital Technium, Swansea University, SA2 8PP, UK, w.newton@swansea.ac.uk
[2] Mustang Marine, The Dockyard, Pembroke Dock, Pembrokeshire, SA72 6TE, UK

Abstract The ventilation of marine vessel engine rooms is important due to energy costs and thermal comfort for personnel. The presented work aims to evaluate the performance of the use of CFD to model a marine vessel engine room in conjunction with field measurements. The field measurements provided data for the boundary conditions and data for comparison with the model. In future work, the model will be used to investigate directed flow, engine room layout and the associated benefits to energy costs and thermal comfort, and improving overall sustainability.

1. Introduction

The ventilation of marine vessel engine rooms is vitally important and has a range of purposes; supplying the air for combustion to the engines, providing the engines with cool air to extract unwanted heat, maintaining the temperature within an adequate range so personnel can work in the engine room [1], and making sure electrical components do not over heat.

Whilst large boat and engine manufacturers, may have advanced engineering facilities with longstanding experience in this area, there is very little published research directly relating the use of CFD to the ventilation of marine engine rooms. There have been publications of research into other applications of CFD to ventilated enclosures with heat transfer, such as, but not limited to, [2]. A presentation by private CFD consultants [3] considers very similar research, but this has not been published in an academic journal.

The purpose of the work presented was to accurately model a marine vessel engine room and compare to field measurements. The impact of computational parameters was investigated to find the effect they had on the solution. The model will be used for a full investigation into, and improvement of, the overall thermal profiles for extreme climates, the use of ducted flow and optimized layout of components for improved efficiencies for sustainability. The performance indicators will be working conditions for personnel and efficiencies of engines due to the intake temperatures.

2. Numerical model

2.1 Physical model

Within the engine room there is a diesel engine for propulsion and a diesel generator set (genset). Air is delivered to the front starboard side of the engine room by means of an axial fan which is situated externally above, whilst the outlet is positioned towards the rear of the room, and is left open to atmospheric conditions. The locations of the inlet and outlet can be seen in Figure 1 along with the engine and genset. The engine has a single intake for air which is purely for combustion, the main cooling for the engine is conducted through a dual heat exchanger using sea water as the final coolant. The exhaust is taken directly out of the room through pipes which are lagged. The genset has a single inlet for combustion air, however, it uses a radiator for cooling. For this, air is drawn over the genset by a fan, and hot air exhausted into the room. The combusted gasses are also taken directly out of the room by lagged exhaust pipes.

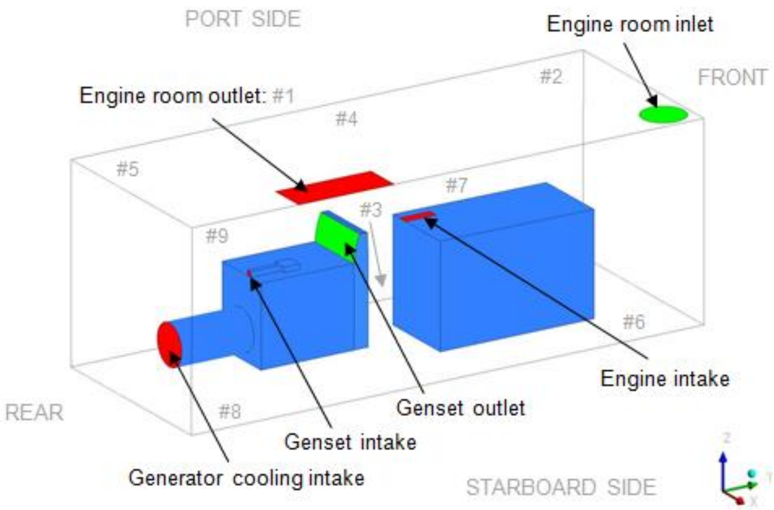

Figure 1. Geometry of engine room including data location numbers

2.2 Boundary conditions, grid and numerical accuracy

The boundary conditions for the numerical model were acquired from the data gathering on the survey vessel. The flow and temperature were recorded for each of the inlets and outlets to the room, the engine and genset. Surface temperatures of the engine and genset were determined by using a thermal imaging camera used in conjunction with surface mounted thermocouples.

FLUENT v14.0 [4] was used to simulate the flow and the heat transfer across the engine room. The governing equations were discretised with a second order upwind scheme, using the SIMPLE algorithm to couple the momentum and pressure equations [5]. The steady state solution was considered converged when the normalised residuals for continuity, momentum and turbulence reached 10^{-3} and energy 10^{-6} respectively.

Four structured grids were used with resolutions ranging from 1,500,000 to 8,000,000 hexahedral elements. Numerical convergence was achieved for each of the grids and the results showed little variation for temperature for all grid resolutions. Considering computational resources, a mesh with a grid size of 1,500,000 elements was used.

3. Results

3.1 Experimental data and validation of model

Temperature data was recorded for the fluid flow at various locations across the engine room, as shown in Figure 1, and was used for comparison with the model. The temperature data recorded in the experiments, and extracted from the numerical model, are compared in Figure 2.

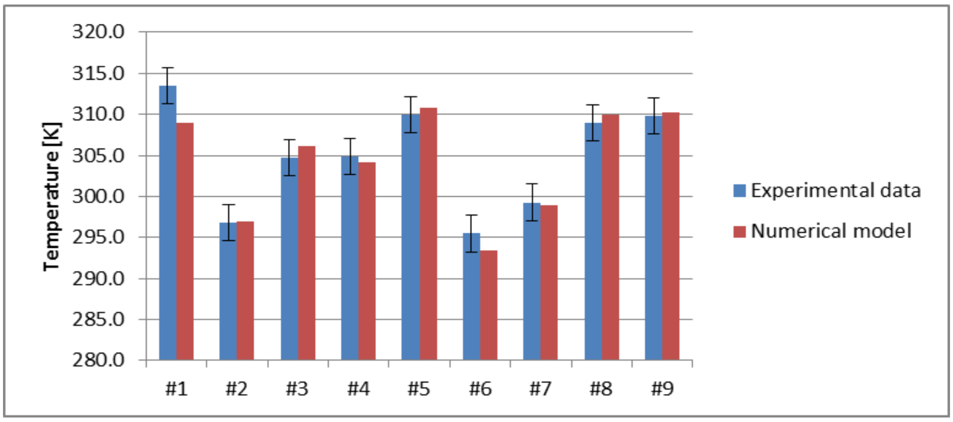

Figure 2. Temperature at locations for experimental data and numerical model.

The numerical results are comparable to the trial data, and qualitatively follow the same trends. The average difference of the numerical and trial data sets was 0.4% with a maximal difference of 1.45% (or 11.29% when considering the Celsius scale). Therefore, it can be concluded that the numerical model accurately represents the physics associated within the engine room [6].

3.2 Temperature profile of room

The temperature profile of the room is represented by a side view in figure 3; a top view shows a similar result. The profile can be split into three sections; the front of the room is cold, in the region of 291.0 K, due to the immediate effect of the cool air coming in from the fan, the middle section is slightly warmer, from 303.4 – 309.6 K, due to the air being heated as it passes the main engine, and the rear of the room is hot with 322.0 K, due to a combination of the air being continually heated by the engine and genset as it travels from the front to the back, plus the addition of hot air being exhausted from the genset. In terms of the thermal profile, this poses a large range of 31 K.

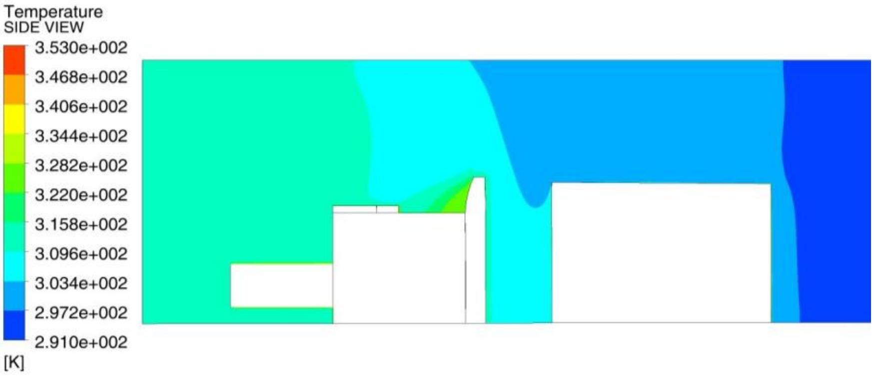

Figure 3. Temperature profile of the engine room; side view

4. Conclusions and future work

A numerical model of a marine vessel engine room has been shown to be comparable to the experimental data with the temperatures at all data points being within 1.45%. The model will be used to analyse the effects of different climatic operating conditions and design studies will be conducted on the engine room inlet to determine whether the engine room and its components can benefit from directing the flow, for a more efficient and sustainable marine vessel.

Acknowledgments

The authors would like to acknowledge the support of the Advanced Sustainable Manufacturing Technologies (ASTUTE) project, which is part funded from the EU's European Regional Development Fund through the Welsh European Funding Office, in enabling the research upon which this paper is based.

References

[1] BS EN ISO 8861:1998. Shipbuilding - Engine-room ventilation in diesel engine ships - Design requirements and basis of calculations, British Standards. (1998)
[2] Rhodin, P and Moshfegh, B. "Numerical modelling of industrial indoor environments: A comparison between different turbulence models and supply systems supported by field measurements" Building and Environment. Vol 46, pp 2365-2374 (2011)
[3] Mustakallio, P and Kosonen, R. "Development of ventilation strategy in diesel engine power plant by using CFD modelling" www.aiha.org/aihce06/handouts/b2mustakallio.pdf (Accessed 2014)
[4] ANSYS software, www.ansys.com (Accessed 2013).
[5] Patankar, S. Numerical Heat Transfer and Fluid Flow. Hemisphere Publishing Corporation, Taylor & Franics Group, New York (1980)
[6] Oberkampf, W. and Trucano, T. Verification and validation in computational fluid dynamics. Progress in Aerospace Sciences. Vol 38, pp. 209-272 (2002)