HEFAT2010
7th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics
19-21 July 2010
Antalya, Turkey

JET IMPINGEMENT HEAT TRANSFER AT HIGH REYNOLDS NUMBERS AND LARGE DENSITY VARIATIONS

Jensen M. V. and Walther J. H.*+
*Author for correspondence
Department of Mechanical Engineering,
Technical University of Denmark,
Nils Koppels Allé, building 403,
DK-2800 Kgs. Lyngby,
Denmark
E-mail: jhwa@mek.dtu.dk

ABSTRACT

Jet impingement heat transfer from a round gas jet to a flat wall has been investigated numerically in a configuration with H/D=2, where H is the distance from the jet inlet to the wall and D is the jet diameter. The jet Reynolds number was 361.000 and the density ratio across the wall boundary layer was 3.3 due to a substantial temperature difference of 1600K between jet and wall. Results are presented which indicate very high heat flux levels and it is demonstrated that the jet inlet turbulence intensity significantly influences the heat transfer results, especially in the stagnation region. The results also show a noticeable difference in the heat transfer predictions when applying different turbulence models. Furthermore calculations were performed to study the effect of applying temperature dependent thermophysical properties versus constant properties and the effect of calculating the gas density from the ideal gas law versus real gas data. In both cases the effect was found to be negligible.

INTRODUCTION

Jet impingement flows provide one of the most efficient ways to transfer energy by convection between a gas and a wall when phase change is not employed. Therefore jet impingement heating or cooling have found widespread use in industrial applications such as material processing and in the manufacturing industry [1]. Jet impingement heat transfer has been investigated intensively over the last four decades, both experimentally and numerically. Some reviews describing the flow physics and proposed heat transfer correlations are [2] and [3]. They treat both round jet and slot jets configurations. Additionally [3] also includes flame jet impingement heat transfer. Although intensively investigated, most researchers have been focused on jet impingement studies with relatively low to moderate jet Reynolds numbers, i.e. generally lower than 100.000. These studies also focused almost exclusively on impingement flows with a relative small temperature difference between jet and wall, as also pointed out in [4], and thereby a small variation in density across the wall boundary layer and in the thermophysical gas properties.

The motivation for the work presented in this paper is rather different from that found in most papers published on jet impingement heat transfer as it is an interest in studying the piston surface heat transfer at flame/wall interaction during combustion in large marine diesel engines. Thus the focus of this work is on high Reynolds number impinging jet flows with large temperature gradients across the wall boundary layer in a high pressure environment in order to represent the conditions during flame impingement in large marine diesel engines.

In this work a hot round turbulent gas jet impinging normally onto a colder flat wall was studied numerically at a very high jet Reynolds number and a large difference between jet and wall temperature using the commercial computational fluid dynamics (CFD) code STAR-CD version 4.10. The local heat flux distribution along the wall was obtained as the main parameter of interest. The heat flux distribution for different jet inlet turbulence intensities was examined as well as the distribution when applying temperature dependent and constant thermophysical properties, respectively. The heat flux distribution was also examined for density calculations based on the ideal gas law and directly on real gas data, respectively. Finally, the distributions predicted when applying four different turbulence models were also studied.

+Also at: Computational Science and Engineering Laboratory, ETH Zürich, Universitätsstrasse 6, CH-8092 Zürich, Switzerland

NOMENCLATURE

c_p	Specific heat capacity at constant pressure
D	Jet inlet diameter
h	Heat transfer coefficient
H	Distance between jet inlet and wall
Nu	Nusselt number, Nu = h D / λ
p	Pressure
q''	Wall heat flux
r	Radial distance from centre axis
Re	Jet Reynolds number, Re = V D ρ / μ
T_f	Film temperature, $T_f = ½ (T_j + T_w)$
T_j	Jet temperature
T_w	Wall temperature
TI	Jet inlet turbulence intensity
V	Jet inlet velocity

Special characters

λ	Thermal conductivity
μ	Molecular viscosity
ρ	Density

PROBLEM DESCRIPTION

The problem investigated was an impinging jet configuration where a heated round turbulent jet impinged normally onto a colder flat wall of constant temperature, see Figure 1. The impinging jet flow can be divided into three characteristic regions [2]: The jet first develops as a free jet in the free jet region where momentum transfer with the surrounding gas broadens up the jet while decreasing the average jet axial velocity. The jet then enters the stagnation region where it due to the presence of the wall is decelerated in the normal direction to the wall and turned into an accelerating flow parallel to the wall. The jet then transforms into a decelerating wall jet in the wall jet region due to momentum transfer across its outer boundary to the surrounding gas and due to momentum exchange with the wall.

This study was initiated due to an interest in investigating the thermal load experienced on the piston surface in large marine diesel engines at flame impingement on the piston during the combustion process. The dimensions of the system as well as the thermophysical conditions were therefore chosen based on relevant dimensions and conditions in large marine diesel engines during combustion. The jet inlet diameter D was 50 mm and the distance H between the jet inlet and the wall was 100 mm giving a H/D ratio of 2. The jet inlet temperature T_j was 2000°C while the wall temperature T_w was fixed at 400°C. The jet inlet velocity V was 10 m/s and the pressure p in the system was 180 bar. The jet as well as the surrounding fluid was atmospheric air. These conditions resulted in a jet Reynolds number (Re) of 361.000 with Re = V D ρ / μ and ρ and μ evaluated at the film temperature $T_f = ½ (T_j + T_w)$. If ρ and μ were evaluated at the jet inlet temperature T_j instead, the jet Reynolds number would be 166.000.

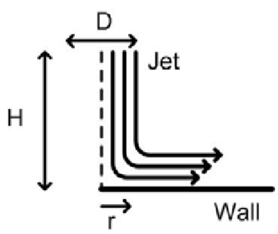

Figure 1 Impinging jet configuration.

NUMERICAL PROCEDURE

The numerical study of the impinging jet problem with the dimensions and themophysical conditions described above was carried out using the commercial CFD code STAR-CD version 4.10. STAR-CD employs the finite volume method and a discretisation up to second order of the governing Navier-Stokes equations, mass, energy and turbulence equations. Details on the numerical study are given below.

Geometry and boundary conditions

The impinging round jet configuration was simulated using a 5 degrees cylindrical sector mesh assuming axis symmetry in the jet configuration. The dimensions of the computational domain were 100 mm x 300mm in the vertical and horizontal directions, respectively. The geometry is shown in Figure 2. All simulations were performed on a cylindrical structured mesh with gradually refined cells in the wall normal direction close to the wall. In the azimuthal direction the grid consisted of only one cell due to the two-dimensionality assumption of the configuration investigated. A no-slip wall boundary condition with a fixed temperature of 400°C was imposed on the bottom of the domain. At the jet inlet an inlet boundary condition was imposed with a plug flow into the domain of 10 m/s and a temperature of 2000°C. The turbulence intensity at the inlet was specified to 5%. This is close to the values reported at the nozzle exit for impinging jet pipe flows in [5] and [6] which were 4.1% and 4%, respectively. The turbulence length scale at the inlet was set to 7% of the jet inlet diameter, i.e. 3.5 mm. At the top and right boundary of the domain pressure boundary conditions were imposed with a static pressure of 180 bars. At the upper pressure boundary the temperature of incoming flow was fixed to 2000°C which equalled the jet inlet temperature. On each side of the 5 degree sector mesh symmetry boundary conditions were imposed to enforce an axis symmetric flow.

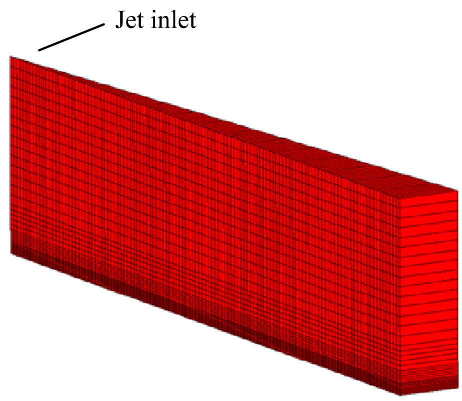

Figure 2 Computational domain.

Turbulence model

For modelling of turbulence the V2F model [7] was chosen which is a RANS type eddy viscosity model. A RANS type eddy viscosity model was selected for two reasons. Firstly because they are less computational expensive than more advanced models like Reynolds stress models and LES models. Secondly, because the aim of the study was an investigation of the time averaged heat transfer in an impinging jet configuration, so instantaneous fluctuating values were not important to resolve. The V2F model does not employ wall functions, so it was necessary to resolve the wall boundary layer. The V2F turbulence model has been shown to be one of the most successful RANS type models for predicting heat transfer in impingement jet configurations, also in the stagnation region where many other RANS type models fail [1,8,9,10]. The V2F model offers a good balance between prediction accuracy and computational costs and has been recommended as the preferred RANS type model for impingement jet heat transfer calculations [1, 8].

Calculations with other turbulence models were also performed for comparison of results with the V2F turbulence model results. The models were a k-ε RNG model employing wall functions instead of resolving the wall boundary layer, a low-Re k-ε model and a low-Re k-ω SST model.

Thermophysical properties and density model

The influence of applying a temperature dependent specific heat capacity c_p, thermal conductivity λ and molecular viscosity μ based on real gas data versus constant properties was studied. The effect of using a temperature dependent density based on the ideal gas law versus a temperature dependent density based directly on real gas data was also investigated.

Polynomial expressions were derived for the temperature dependency of the properties and density based on real gas data for a constant pressure of 180 bar using data in [11]. If not otherwise stated the presented results were obtained using temperature dependent properties and a density based on real gas data.

Convergence check

The computations showed a need for a large number of iterations (usually more than 40.000) before complete convergence was obtained, i.e. the solution did not change with further iterations. This is thought to be partly due to imposing pressure boundary conditions on large portions of the computational domain surface, and party due to the presence of very small cells near the wall in order to resolve the wall boundary layer. The central difference scheme was used to discretise the momentum, mass, energy and turbulence equations, which can also have been contributing to increasing the need for iterations before convergence was obtained. The sufficient total residual tolerance for obtaining complete convergence in the computations was generally 10^{-7}.

RESULTS AND DISCUSSION

Results for the heat flux distribution along the wall are presented in terms of the Nusselt number (Nu) distribution. The Nusselt number is calculated as $Nu = h D / \lambda$ where the heat transfer coefficient h is determined as $h = q'' / (T_j - T_w)$ with q'' being the wall heat flux. λ is evaluated at the film temperature T_f.

Grid independency and domain extension checks

A mesh sensitivity test was performed using two grids where cell sizes in the refined grid were half of the cell sizes in the reference grid in both radial and horizontal directions. As the simulations were two-dimensional no grid refinement was performed in the azimuthal direction. The reference grid consisted of 108.000 cells and the refined grid of 432.000 cells. The results are shown in Figure 3. Dividing the cells in the reference grid by a factor of two in both the horizontal and radial direction only influenced slightly the results: In the stagnation region, which was of main interest in this study, the curves were coinciding; at larger radii a small discrepancy between the curves was observed. The maximum deviation in the Nusselt number distribution was 2.5%, whereas it was less than 0.5% in the stagnation region. As an additional check the wall shear stress distribution was also investigated, although it is not shown here. The maximum deviation for the wall shear stress was 4% at r/D=6. Based on these results, the reference grid was considered to provide sufficient resolution of the computational domain and was used for the further simulations. The highest y+ value of the near wall cells using the reference grid was 0.12.

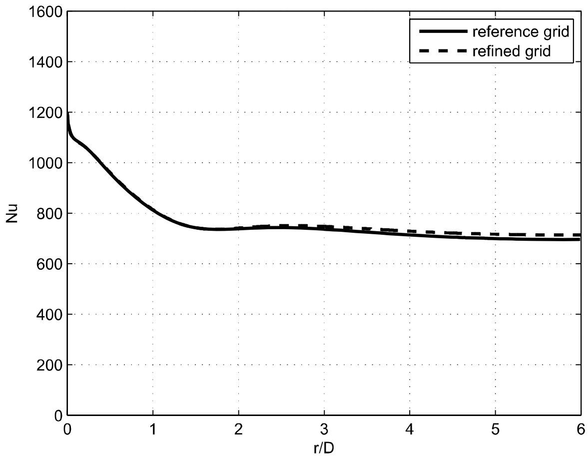

Figure 3 Nusselt number distribution for grid independency check.

Heat transfer results are shown in Figure 4 for a simulation performed on a grid with the same refinement as the reference grid, but extended in radial direction so the maximum r/D was 10. Extending the domain in radial direction, and thus moving the pressure boundary further out, did not influence the heat transfer distribution or shear stress distribution (not shown here) significantly. Especially in the stagnation region the influence seemed very small. For a dimensionless radial distance greater than 4, however, a slight discrepancy could be observed between the curves. As this discrepancy was considered insignificant, and due to increased computational costs with increasing domain size, the reference grid with a maximum r/D of 6 was found sufficient for this work. It has previously been reported that once the domain was extended more than r/D=8+H/D there was no noticeable effect on the flow field and local heat transfer results [8,1].

A study of the influence of extending the computational domain vertically upwards by moving the upper pressure boundary further away from the wall was not done here. The reason was some considerations on keeping a domain which in an easy manner could be used in future work to do simulations with an inclined jet. Probably a vertical extension of the domain would to some degree influence the results.

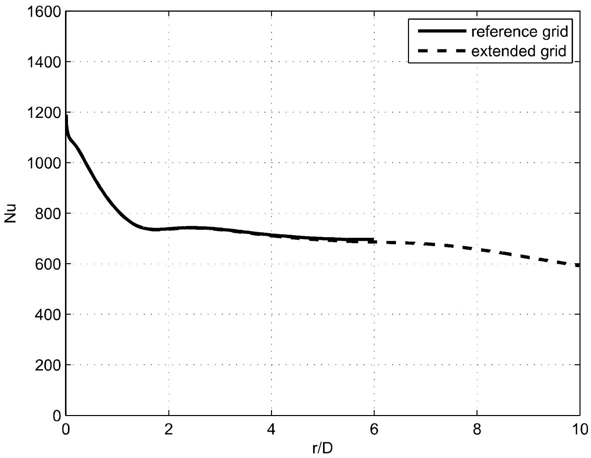

Figure 4 Nusselt number distribution for extended grid check.

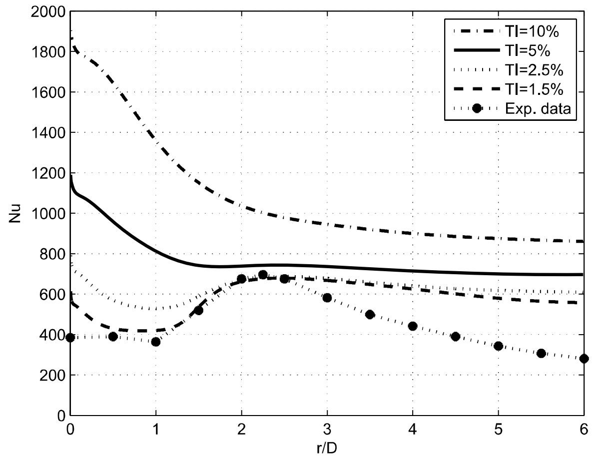

Figure 5 Nusselt number distribution for varying degrees of jet inlet turbulence intensities.

Influence of jet inlet turbulence intensity

The influence of jet inlet turbulence intensity (TI) on the wall heat transfer was investigated by varying the turbulence intensity at the jet inlet boundary from 1.5% to 10%, see Figure 5. It was observed that jet turbulence intensity had a significant influence on the wall heat transfer. The maximum Nusselt numbers attained a value of 680 for a jet inlet turbulence intensity of TI=1.5% and 1900 for TI=10%. A secondary peak in the Nusselt number distribution was clearly observed around r/D=2.5 for low turbulence intensities, i.e. for TI=1.5% and TI=2.5%. For TI=5% a secondary peak was also observed although it was weak. In the case of TI=1.5% the maximum Nusselt number occurred at the secondary peak, whereas for the other cases it was located at the stagnation point. For TI=10% the Nusselt number values decreased monotonically from the stagnation point without any visible secondary peak.

The appearance of a secondary Nusselt number peak has also been reported in previous experimental works, e.g. [6,12,13], where the location of the peak ranged from r/D=2.0 to r/D=2.25 possibly due to different Reynolds numbers. The secondary peak is generally believed to be caused by a transition from a laminar boundary layer flow to a turbulent boundary layer flow, see e.g. [2,10]. However, some authors [8,14] suggested an alternative explanation according to which the stagnation region flow is already turbulent and that the peak is not caused by a laminar-to-turbulent transition but by an augmentation of turbulence kinetic energy due to high shear in the region of streamline convergence.

The strong influence of the turbulence intensity on the wall heat transfer emphasizes the importance of knowing this parameter when comparing different experimental measurements or when comparing with numerical results as was also pointed out in [14].

Experimental data from [13] are also shown in Figure 5 for comparison. The data were obtained in an impinging jet configuration with a Reynolds number of 375.000 and H/D=2. Contrary to the numerical calculations reported in this paper, the experimental investigation focused on a heated plate cooled by an air jet at room temperature and atmospheric conditions. The turbulence intensity at the nozzle exit in the experiment was reported to TI=0.11%. The experimental data deviate from the numerical results in the stagnation region and in the fully developed wall jet region, but in the area of the secondary peak the experimental data and the numerical values for the case of a jet inlet turbulence intensity of TI=1.5% show a very good agreement. The reason for the deviation in the stagnation region is thought to be caused by the difference in jet inlet turbulence intensity between the numerical and the experimental configurations. Unfortunately it was not possible to obtain converged results with a lower jet inlet turbulence intensity than TI=1.5%. The deviation in the fully developed wall jet region may be assigned to the fact that the density variation across the wall boundary layer in the numerical configuration was much larger than in the performed experiment. Another reason for the deviations could be that the plate was cooled by the jet in the experiment while the wall was heated by the jet in the numerical investigation. Unfortunately the authors did not find other experimental data for comparison where the physical conditions were closer to that imposed in the numerical calculations than in [13]. It would be of interest to obtain such data for a more detailed validation of the numerical results.

Figure 6 shows calculated stagnation point Nusselt numbers versus jet inlet turbulence intensity. An approximate linear relationship is observed between the stagnation point Nusselt number and the turbulence intensity. If the linear relationship is extrapolated to a turbulence intensity of TI=0.11%, a stagnation point Nusselt number of 425 is obtained which is in good agreement with the experimentally measured stagnation point Nusselt number of 384 reported in [13] and shown in Figure 5.

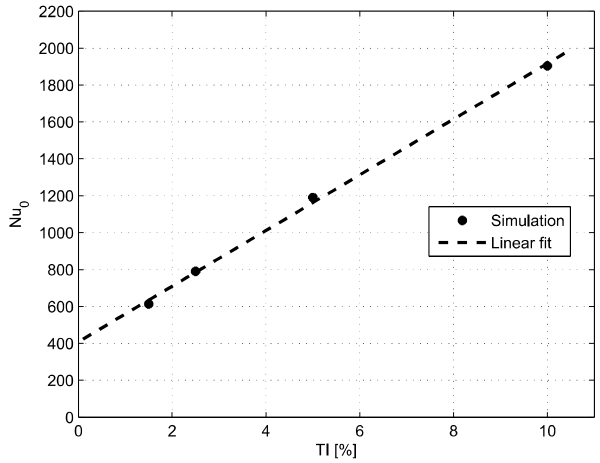

Figure 6 Stagnation point Nusselt number versus jet inlet turbulence intensity.

Influence of temperature dependent properties and density model

The influence of applying temperature dependent properties (c_p, λ and μ) based on real gas data versus constant properties was investigated. This was of interest because the temperature span in the numerical configuration from wall to jet inlet was 1600K. This caused c_p to vary by 24%, μ by 124% and λ by 187% in the computational domain. The effect of using a temperature dependent density based on the ideal gas law versus real gas data was studied as well. These investigations resulted in four cases for which the results are shown in Figure 7. In the cases where the thermophysical properties where kept constant the properties were evaluated at T_f. Cases where the density was kept constant were also investigated but it was not possible to obtain convergence in those calculations so they are not included here.

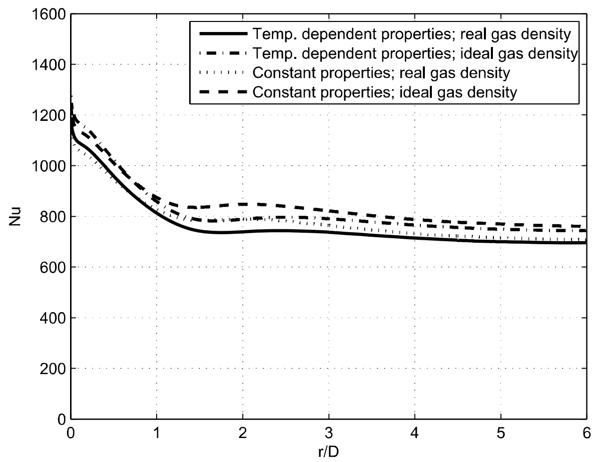

Figure 7 Nusselt number distribution for calculations applying constant and temperature dependent thermophysical properties and different density models.

Using the ideal gas model to calculate the density instead of using real gas data increased the predicted Nusselt number values approximately 7% along the wall, both in the case of temperature dependent properties and in the case of constant properties. Keeping the thermophysical properties constant increased the predicted Nusselt number values 2-6% along most of the wall compared to the case with temperature dependent properties. A minor decrease was observed in the stagnation region. This trend was seen both when the density was based on the ideal gas model and when calculating it from real gas data. Using a density based on the ideal gas law and keeping the properties constant at the same time, increased the Nusselt number values 10% in average compared to the case where the properties were temperature dependent and the density was based on real gas data. Thus if heat transfer calculations for conditions similar to those studied in this work are performed mainly for the purpose of observing trends in the predictions, for instance while making parameter variations, it may be sufficient to use constant properties and a density based on the ideal gas law.

Influence of turbulence model

All previous presented results were obtained using the V2F turbulence model. Calculations were performed with three other well known turbulence models in order to observe variations in the heat transfer predictions resulting from application of different turbulence models. The three models, which are all described in [15], were a low-Re k-ε model [16], a k-ω SST model [17] and a k-ε RNG model [18] which employed standard wall functions [19]. The first two models needed a resolved wall boundary layer while the last model was cheaper to use due to the application of wall functions.

The low-Re k-ε model and the k-ω SST model were applied on the same grid of 108.000 cells as the previous calculations. The k-ε RNG model with wall functions was applied on a coarser grid with 4200 cells as it was not necessary to resolve the wall boundary layer for this model which led to y+ values between 30 and 65. The computation times showed that the calculations using the k-ε RNG model obtained convergence an order of magnitude faster than the calculations using the other models. Figure 8 shows the obtained heat transfer results.

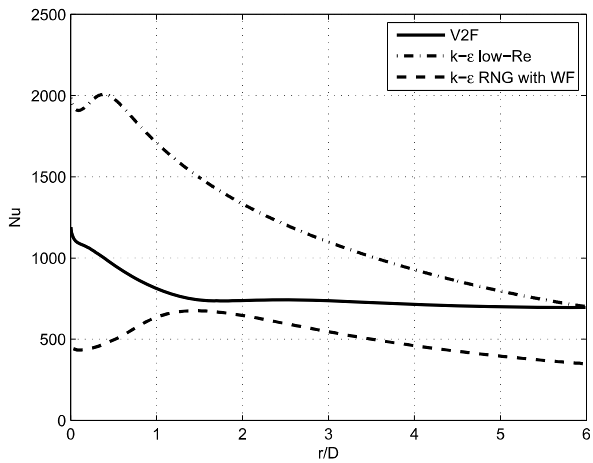

Figure 8 Nusselt number distribution obtained using different turbulence models.

A large scatter in the predicted heat transfer results was observed when applying the different turbulence models, both in magnitudes and trends. The Nusselt number predictions using the V2F model was first decaying until r/D=1.7 whereafter a local maximum was seen at r/D=2.5. The global maximum was at the stagnation point. For the two k-ε models the Nusselt number predictions showed a local minimum much closer to the stagnation point, already at r/D=0.1 and a secondary peak at r/D=1.5 and r/D=0.4, respectively. In both cases the secondary peak was also the global maximum in contrast to the V2F predictions. The magnitudes of the Nusselt number values differed also greatly between the different model predictions, especially in the stagnation region. Most pronounced were the low-Re k-ε model predictions. The results obtained applying the k-ω SST model did converge but the flow field was not realistic. It showed an additional jet-like inflow just next to the specified jet inflow with an even higher flow velocity than that specified for the jet itself. Therefore the k-ω SST model results are not included here.

In summary the results in Figure 8 emphasize the problem of handling impinging jet flows for the turbulence models, especially in the stagnation region under the given severe simulation conditions.

CONCLUSIONS

An impinging jet configuration characterized by a high jet Reynolds number and a large density variation across the wall boundary layer was studied numerically with focus on wall heat transfer predictions. The configuration conditions represented the conditions during combustion in large marine diesel engines. The maximum Nusselt number reached a value of 1200 which was in the stagnation point of the jet impingement.

The jet inlet turbulence intensity was found to have a pronounced influence on the heat transfer prediction, especially in the stagnation region which is consistent with previous investigations. Thus it is highly important in jet impingement heat transfer experiments to document the jet inlet turbulence level for the purpose of comparison with other investigations.

The influence of applying temperature dependent thermophysical properties versus constant properties was investigated as well as the effect of calculating the density from the ideal gas law instead of using real gas data. It was found that these variations changed the heat transfer predictions less than 10% in average, despite the large temperature span of 1600K in the numerical configuration. It may therefore be sufficient in many applications to apply constant properties and base the density on the ideal gas law.

Applying different turbulence models gave a large scatter in the heat transfer predictions, especially in the stagnation region. Thus it is important to apply a turbulence model which has been shown to achieve good heat transfer predictions in impinging jet configurations.

Furthermore, it was found that previous studies, both experimental and numerical ones, on impinging jet heat transfer seemed quite limited in the cases of high jet Reynolds numbers and for large temperature span in the configuration, although studies on impinging jet flows in general are numerous. Further investigations in this field of impinging jet heat transfer would be appreciated, especially experimental work which would be very valuable for validation of numerical results.

ACKNOWLEDGEMENTS

This work was financially supported by MAN Diesel and the Danish Center for Applied Mathematics and Mechanics (DCAMM).

REFERENCES

[1] Zuckerman, N. and Lior, N., Impingement heat transfer: Correlations and numerical modeling, *J. Heat Transfer,* Vol. 127, 2005, pp. 544-552.

[2] Marin, H., Heat and mass transfer between impinging gas jets and solid surfaces, *Adv. Heat Transfer,* Vol. 13, 1977, pp. 1-60.

[3] Viskanta, R., Heat transfer to impinging isothermal gas and flame jets, *Exp. Therm. Fluid Sci.*, Vol. 6, 1993, pp. 111-134.

[4] Shi, Y., Mujumdar, A. S. and Ray, M. B., Effect of large temperature difference on impingement heat transfer under a round turbulent jet, *Int. Comm. Heat Mass Transfer,* Vol. 31, 2004, pp. 251-260.

[5] Baughn, J. W., Hechanova, A. E. and Yan, X., An experimental study of entrainment effects on the heat transfer from a flat surface to a heated circular impinging jet, *J. Heat Transfer,* Vol. 113, 1991, pp. 1023-1025.

[6] Hofmann, H. M., Kind, M. and Martin, H., Measurements on steady state heat transfer and flow structure and new correlations for heat and mass transfer in submerged impinging jets, *Int. J. Heat Mass Transfer,* Vol. 50, 2007, pp. 3957-3965.

[7] Durbin, P. A., Near-wall turbulence closure modeling without "damping functions", *Theoret. Comput. Fluid Dynamics*, Vol. 3, 1991, pp. 1-13.

[8] Behnia, M., Parneix, S. and Durbin, P. A., Prediction of heat transfer in an axisymmetric turbulent jet impinging on a flat plate, *Int. J. Heat Mass Transfer,* Vol. 41, 1998, pp. 1845-1855.

[9] Hofmann, H. M., Kaiser, R., Kind, M. and Martin, H., Calculations of steady and pulsating impinging jets - an assessment of 13 widely used turbulence models, *Numerical Heat Transfer, Part B,* Vol. 51, 2007, pp. 565-583.

[10] Gordeev, S., Heinzel, V. and Slobodchuk, V., Simulation of single and multiple impinging jet cooling and comparison with experimental data, *Proc. 8th Biennial ASME Conf. on Engineering Systems Design and Analysis (ESDA2006),* 2006, pp. 143-158.

[11] Perry, R. H., Green, D. W. and Maloney, J. O., Perry's chemical engineers' handbook, Sixth edition, *McGraw-Hill*, 1984.

[12] Baughn, J. W. and Shimizu, S., Heat transfer measurements from a surface with uniform heat flux and an impinging jet, *J. Heat Transfer,* Vol. 111, 1989, pp. 1096-1098.

[13] Petzold, K., Der Wärmeübergang an einer senkrecht angeströmten Platte, *Wissenschaftliche Zeitschrift der technischen Universität Dresden*, Vol. 13, 1964, pp. 1157-1161.

[14] Behnia, M., Parneix, S., Shabany, Y. and Durbin, P. A., Numerical study of turbulent heat transfer in confined and unconfined impinging jets, *Int. J. Heat Fluid Flow,* Vol. 20, 1999, pp. 1-9.

[15] STAR-CD version 4.10, Methodology, CD-adapco, 2009.

[16] Lien, F.S., Chen, W.L. and Leschziner, M.A., Low-Reynolds-Number Eddy-Viscosity Modelling Based on Non-Linear Stress-Strain/Vorticity Relations, *Proc. 3rd Symp. on Engineering Turbulence Modelling and Measurements, Crete, Greece*, 1996.

[17] Menter, F.R., Zonal two equation *k*-ω turbulence models for aerodynamic flows, *Proc. 24th Fluid Dynamics Conf., Orlando, Florida, USA, 6-9 July,* Paper No. AIAA 93-2906, 1993.

[18] Yakhot, V., Orszag, S.A., Thangam, S., Gatski, T.B. and Speziale, C.G., Development of turbulence models for shear flows by a double expansion technique, *Phys. Fluids A – Fluid Dynamics*, Vol. 4, 1992, pp. 1510–1520.

[19] Launder, B. E. and Spalding, D. B., The numerical computation of turbulent flows, *Comp. Meth. Appl. Mech. & Engng.,* Vol. 3, 1974, pp. 269-289.