2
DOE/AL/33183--T5

DE88 003398

DOE/AL/33183--T5
Contract DE-AC04-86AL33183

Technical Progress Report

MULTIDIMENSIONAL MODELING OF CONVECTIVE
HEAT TRANSFER WITH APPLICATION TO I.C. ENGINES

Mark J. Jennings and Thomas Morel

Integral Technologies Incorporated
645 Blackhawk Drive
Westmont, Illinois 60559

Prepared for:

Department of Energy
Albuquerque Operations Office
P.O. Box 5400
Albuquerque, New Mexico 87115

Date: December 11, 1987

DISCLAIMER

This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, nor any of their employees, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof. The views and opinions of authors expressed herein do not necessarily state or reflect those of the United States Government or any agency thereof.

Objectives/Scope

The objective of this project is to develop a more comprehensive understanding of the convective heat transfer process in complex, unsteady turbulent reacting flows, typical of those which occur in internal combustion engines. The specific area of research is the representation of heat transfer in detailed multi-dimensional Navier-Stokes models, and modeling of turbulent transport mechanisms. The detailed tasks include a review of relevant prior work. Based on this review, and original work done under this contract, several modeling approaches will be formulated and further studied and tested. The tests will be carried out on flow cases which have relevance to engine flows, and for which reliable experimental data exist. Such data will be sought and identified. The analytical studies will lead to the determination of the best modeling approaches to be used for heat transfer simulation in internal combustion

MASTER

DISTRIBUTION OF THIS DOCUMENT IS UNLIMITED

DISCLAIMER

This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency Thereof, nor any of their employees, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof. The views and opinions of authors expressed herein do not necessarily state or reflect those of the United States Government or any agency thereof.

DISCLAIMER

Portions of this document may be illegible in electronic image products. Images are produced from the best available original document.

engines. Following that, a detailed study will be carried out of spatial and temporal heat flux distribution in a representative engine. This will be complemented by a parametric study of engine heat transfer dependence on intake flow details, combustion chamber geometry, engine speed and engine load.

Technical Highlights

The treatment of wall boundary conditions is critical to the accurate calculation of convective heat transfer. In multi-dimensional engine flow codes, wall functions are applied to bridge the gap between the solution grid and the combustion chamber boundaries since the near wall boundary layers cannot be properly resolved without the use of excessively fine meshes. These functions are derived from simplified near wall flow analysis and have been developed primarily for the calculation of two-dimensional boundary layer type flows. The basic formulations developed by researchers for these simple flows have been employed in engine flow codes with little or no modification. An investigation of wall modeling techniques was begun with the purpose of determining the accuracy of wall models currently being used in the calculation of engine and non-engine flows. The results of this investigation will lead to wall model improvements and a complete understanding of the strengths and limitations of models currently in use.

The standard approach used by most researchers to develop wall functions has been to assume that the near wall flow consists of two zones: a viscous-stress dominated zone adjacent to the wall in which the turbulent stresses are zero, and an outer, fully turbulent zone in which the viscous stresses are negligible. The commonly assumed mean velocity and temperature profiles within the two regions are illustrated in Figure 1 as a function of wall coordinates. Assuming analogy between turbulent heat and momentum transfer, the two profiles (velocity and temperature) are identical for molecular and turbulent Prandtl numbers of 1.0. The viscous layer is characterized by a linear velocity/temperature profile while the turbulent layer is characterized by a logarithmic velocity/temperature

profile. These profiles are used to determine the near wall heat transfer and shear stress. In addition to providing wall boundary conditions for the momentum and energy equations, the wall functions provide boundary conditions for the turbulence model equations as well. The focus of this report will be on wall shear and heat flux models while future work will be concerned with boundary conditions for the turbulence equations. The latter conditions are of importance, since calculation of near wall heat and momentum transfer requires the determination of near wall turbulence behavior.

In order to facilitate the straightforward testing of wall models, portions of EPISO responsible for handling wall boundary conditions were extensively recoded. Considerable time and effort was spent to verify that all changes were done properly so that the integrity of the code was preserved. The simulation program is currently structured to allow selection of different wall shear, wall heat flux and wall turbulence functions through the input data file. This approach provides the utmost flexibility for consistent testing of wall modeling approaches.

Simulations of fully developed pipe flow without heat transfer were performed as a starting point for the analysis of different wall shear stress model formulations. The pipe flow provides a simple and well-controlled test case that is useful for verifying the correct implementation of models and understanding fundamental differences between models. Calculations were done using three different wall shear model formulations in conjunction with different locations of the node nearest the wall. The different wall shear models tested included a model used by EPISO, a model used by the multi-dimensional engine flow simulation code developed at Los Alamos National Laboratory known as KIVA and a model developed by Chieng and Launder, 1980. The EPISO and Launder models, hereafter referred to as model E and model C, respectively, both relate the wall shear stress directly to the near wall turbulence level. Model E uses the turbulence level at the near wall node, while model C uses the turbulence level at the viscous layer-turbulent layer interface. The KIVA model, hereafter referred to as model K, is derived assuming a $1/7$ power law

relation for the near wall mean velocity. The resulting expression for the wall shear does not directly depend on the near wall turbulence level. These differences have important consequences regarding heat transfer (as well as skin friction) calculations, since nearly all heat flux models employ Reynolds analogy between momentum and heat transfer to determine the heat flux from the wall shear stress.

Predictions of the fully-developed mean velocity profiles for the three models are shown in Figure 2 as a function of the near wall coordinate, y^+ . The predictions of the three models are very nearly identical, with differences arising primarily from slight differences in the friction velocity used to nondimensionalize the mean velocity. The invariance of the velocity profile results is not unexpected, since all of the models are consistent with the law of the wall, which is satisfied in fully developed pipe flow. However, significant differences between the models will certainly arise in more complex flows. The calculated mean velocity profiles follow the log profile out to $y^+ = 200$.

Calculations were performed for y^+ locations of the near wall node point equal to 7, 20 and 70. The predicted mean velocity profiles are shown in Figure 3. As can be seen from the figure, moving the near wall node location within the fully turbulent layer had little effect on the mean velocity profile. This is to be expected since the log layer extends over the whole region and all of the models were designed to take advantage of this fact. However, placing the first node in the viscous sublayer ($y^+ = 7$) generated significant discrepancies in the predicted mean velocity profile. This is caused by large errors in the predicted wall shear as illustrated in Figure 4 which shows the predicted turbulent shear stress profile across the pipe on a linear scale. The predicted wall shear is close to 20% higher than results obtained with the first node in the fully turbulent region. The higher wall shear resulted in the generation of much higher turbulence levels within the pipe, as illustrated in Figure 5.

These results demonstrate inconsistencies in the two-zone wall layer model. The model has discontinuities in both the eddy diffusivity and mean

velocity gradient across the boundary separating the viscous and fully turbulent layers. As a result, the shear stress is discontinuous across this interface as well. Placing the near wall node in the viscous region brings these discontinuities into the calculation. While in the case of a simple pipe flow calculation, the mesh can be set up so that the near wall node is never in the viscous region, the same cannot be done for an engine calculation due to the complex nature of the flow. Therefore, accurate modeling of the viscous inner region is important and cannot be neglected. A buffer layer model that smoothly merges the viscous layer linear profile with the outer turbulent log profile is required in order to remove the discontinuities in momentum flux across the wall layer. This would provide a more physically realistic representation of the wall layer and would make the predictions insensitive to near wall node location within the layer.

The investigation of wall heat flux models has just been recently started. Calculations were performed using two different wall heat flux models in a motoring, pancake geometry engine at 1500 RPM. The two models considered were the heat flux model used in EPISO and the heat flux model used by KIVA. The EPISO model relates the heat transfer coefficient directly to the near wall turbulence level, while the KIVA heat transfer coefficient depends only on the local mean velocity. As a result, the KIVA model predicts zero heat transfer where there is no mean flow and does not account for heat transfer due to the turbulent fluid motions. Predictions of average heat flux into the cylinder head and liner for the two models are shown in Figures 6 and 7. As can be seen, significant differences exist due to the turbulence effect discussed above. The flow is characterized by low near wall mean flow, and high near wall turbulence levels which brings into play the differences between the models: the KIVA model only reflects the low mean flow effect on heat transfer, while the EPISO model incorporates the effect of the near wall turbulence. This makes the latter model more plausible, but whether or not it reflects the correct magnitude of the effects of turbulence needs to be determined from further analysis. These results highlight some of the major differences which exist between near wall heat transfer models. In the future, calculations will be performed of a developing thermal boundary layer in

fully developed pipe flow using the two heat flux wall models described above, plus one or two additional existing models.

Future Activities

Future work will concentrate on continuing the wall model investigation. Calculations of pipe flow with heat transfer will be performed in order to study the performance of wall heat flux models. Following this, calculations of pipe flow using different $k-\epsilon$ wall boundary conditions will be conducted. After completion of the pipe flow calculations, more complex flow configurations will be considered which will further bring out the differences in the wall function approaches.

References

Chieng, C. C. and Launder, B. E., (1980), "On the Calculation of Turbulent Heat Transport Downstream from an Abrupt Pipe Expansion," Numerical Heat Transfer, Vol. 3, pp. 189-207.

Status of Milestones

Work progressed on the setup of test codes and model testing, Milestones c and d, respectively.

Communications/Visits/Travel

Presentation was given at the 1987 SAE Fuels and Lubricants meeting in Toronto, Canada on November 2-6, 1987.

Problems Encountered

None.

Publications

Jennings, M.J. and Morel, T., "Modeling of Turbulent Heat Transfer with Application to IC Engines," SAE Paper 872104, Toronto, November, 1987.

Figure 1. Wall layer mean velocity, U^+ , and mean temperature, T^+ , profiles for the two zone model. $U^+ = U/u_\tau$, $T^+ = (h_w - h)/(q_w/\rho u_\tau)$ and $y^+ = U_\tau y/\nu$ where U = mean velocity, T = mean temperature, u_τ = friction velocity, ρ = fluid density, y = normal distance from wall, ν = kinematic viscosity, h = enthalpy and q_w = wall heat flux.

Figure 2. Comparison of wall layer mean velocities calculated from three different wall shear models for fully developed pipe flow.

Figure 3. Comparison of wall layer mean velocity profiles for three different near wall node locations in fully developed pipe flow.

Figure 4. Comparison of shear stress profiles for three different near wall node locations in fully developed pipe flow.

Figure 5. Comparison of turbulent kinetic energy profiles for three different near wall node locations in fully developed pipe flow.

Figure 6. Comparison of average heat flux into the cylinder calculated using two different wall heat flux models for motoring, pancake geometry engine. Model 1 refers to EPISO's wall model and Model 2 refers to the KIVA wall model.

Figure 7. Comparison of average heat flux into the cylinder liner calculated using two different wall heat flux models for a motoring, pancake geometry engine. Model 1 refers to EPISO's wall model and Model 2 refers to the KIVA wall model.