

UNCLASSIFIED

Defense Technical Information Center
Compilation Part Notice

ADP012121

TITLE: Modeling Diesel Engine Injector Flows

DISTRIBUTION: Approved for public release, distribution unlimited

This paper is part of the following report:

TITLE: Army Research Office and Air Force Office of Scientific Research.
Contractors' Meeting in Chemical Propulsion [2001] Held in the University
of Southern California on June 18-19, 2001

To order the complete compilation report, use: ADA401046

The component part is provided here to allow users access to individually authored sections
of proceedings, annals, symposia, etc. However, the component should be considered within
the context of the overall compilation report and not as a stand-alone technical report.

The following component part numbers comprise the compilation report:
ADP012092 thru ADP012132

UNCLASSIFIED

MODELING DIESEL ENGINE INJECTOR FLOWS

ARO Contract Number DAAG55-98-1-0318

Stephen D. Heister and Gregory A. Blaisdell

Purdue University, School of Aeronautics and Astronautics
1282 Grissom Hall, W. Lafayette, IN 47907-1282

SUMMARY/OVERVIEW:

This project is focused on the study of flows inside diesel injector flow passages with emphasis on resolution of time-dependent cavitation regions within the device. During the past year, the turbulent axisymmetric model has undergone substantial validation. In addition, an improved inflow boundary condition has been developed to increase accuracy in predicting discharge characteristics of these devices.

TECHNICAL DISCUSSION

A drilled orifice is widely used to provide a simple solution for atomizing liquids and the presence of cavitation inside such a nozzle has received much attention in the past due to its broad applications. Recent experiments have shown that the turbulence in the nozzle hole resulting from cavitation is a mechanism that promotes atomization. Gopalan and Katz^[1] observed that the unsteady cavity collapse involves substantial increases in turbulence intensity, and momentum and displacement thicknesses in the boundary layer. They also showed that the collapse of bubbles is the dominant source of vorticity downstream of a cavity. The present calculations add $k - \omega$ turbulence model to Chen and Heister's^[2] homogeneous fluid model to simulate the turbulent cavitating flow in a nozzle hole. We chose to use the $k - \omega$ model because it has better performance than other turbulence models in adverse pressure gradient flows which are common cases in cavitating flows.

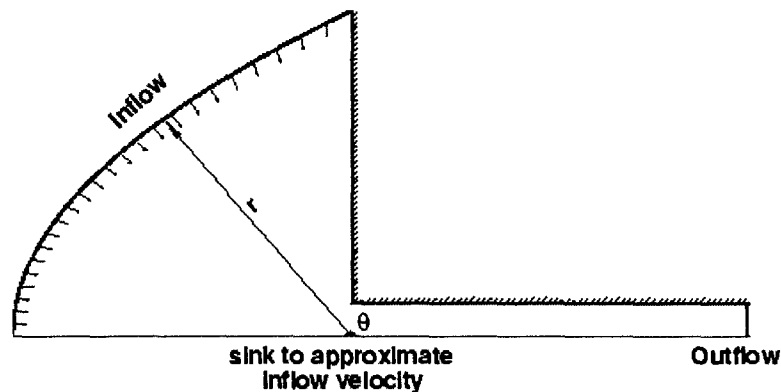


Figure 1 Schematic representation of a sink approximation of inflow velocity.

In flow velocity boundary conditions are obtained by placing an artificial sink at the origin as shown in Fig. 1. For a 2-D problem the velocity at the inflow boundary is calculated by $u_{in} = -(\Lambda / 2\pi r) \cos \theta$, $v_{in} = -(\Lambda / 2\pi r) \sin \theta$. The strength of the sink is updated during each time step by the conservation of mass flow rate through the nozzle passage. The only difference between an axisymmetric problem and a 2-D problem at this point is that a three-dimensional sink is utilized for the former. A few axisymmetric runs were made to compare with the discharge coefficient, C_D , measurements by Nurick^[3] on a circular orifice. Fig. 2 shows the

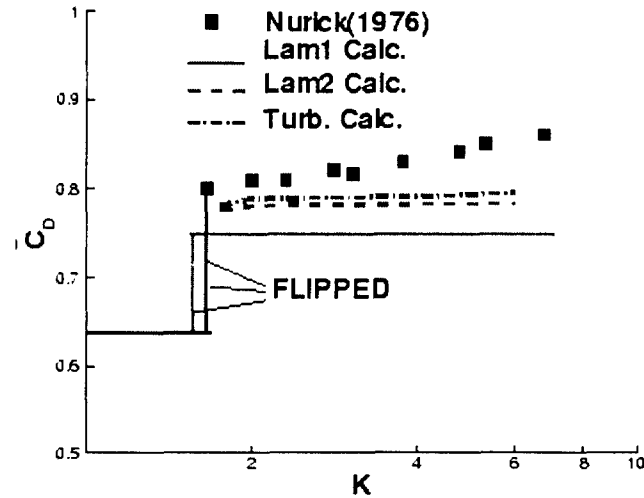


Figure 2 Discharge coefficient C_D comparison with experimental results; $L/D = 6$, $D = 3.18$ mm.

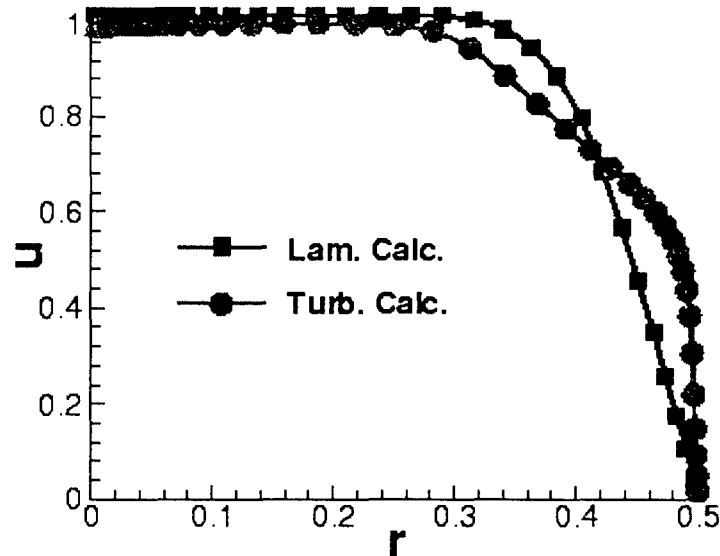


Figure 3 Comparison of velocity profiles at exit for laminar and turbulent solutions $L/D = 6$, $D = 3.18$ mm.

results of a comparison on an orifice of $L/D = 6$, $D = 3.18$ mm, under a back pressure of $P_2 = 13.8$ psi. As shown in Fig. 2, the sink inflow velocity treatment (Lam 2) greatly improves the prediction of discharge coefficient. The turbulence model shows a further improvement and gives results somewhat closer to the experimental data. The differences between the turbulence model and laminar predictions on C_D might be explained by Fig. 3 in which the exit velocity profiles are plotted for both laminar and turbulent calculations. The turbulent velocity profile is fuller than the laminar one. Due to this feature, the turbulence model yields a larger C_D by 1.6%.

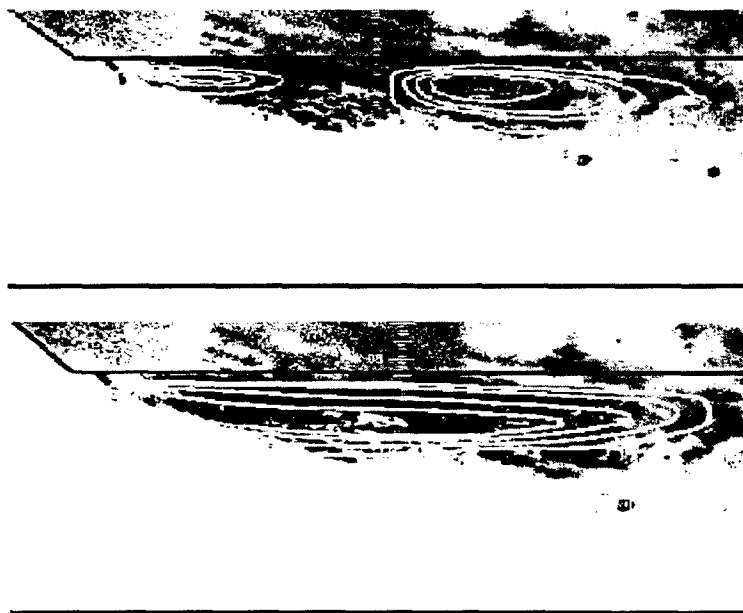


Figure 4 Side-View overlay of cavitation field and predicted pseudo-density contours. The upper image provides a comparison with a laminar flow calculation while the lower image depicts the improved results with a turbulent simulation.

Fig. 4 shows density contours, which denotes the cavity region, obtained from laminar (the upper one) and turbulent (the lower one) calculations, overlaying one photographic snapshot by Henry.^[4] Although the laminar simulation results in an overall cavitation extent consistent with experiment, it indicates two separate regions of cavitation. The turbulent model improves on this point by generating a single cavitation region which appears to be quite consistent with experimental results both in axial and cross-stream directions.

Fig. 5 shows velocity profiles inside the location of the cavity region for cavitating and non-cavitating conditions in a 2-D slot. The flow inside the cavitation region in general is slower under cavitating conditions than under non-cavitating conditions. Especially in the middle of the cavity ($x = 2$) a strong reverse flow occurs between the wall and cavity. At the end of the cavitation region ($x = 3$) the velocity profiles approach the same shape for both cavitating and non-cavitating conditions. Cavitation has a significant effect on the boundary layer development downstream. Although the turbulence model predicts the boundary layer thickness is almost identical for both cavitating and non-cavitating conditions, it produces increased displacement and momentum thicknesses for cavitating conditions. Other researchers^[1] have observed similar behavior experimentally.

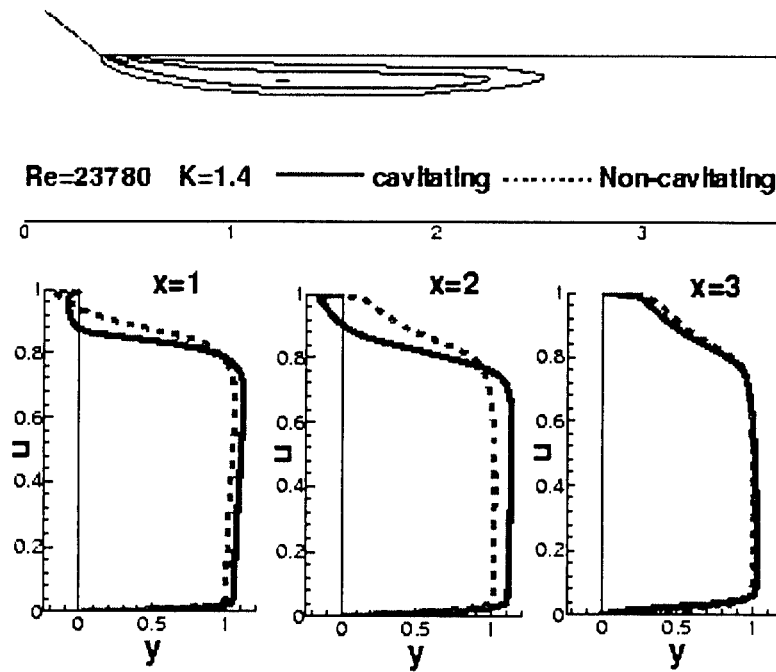


Figure 5 Velocity profiles under cavitating and non-cavitating conditions, $Re = 23780$, $K = 1.4$.

CURRENT EFFORTS

Most of the above results were published in.^[5] Current efforts are aimed at completing the work to be done under this grant. Turbulence model calculations are being done of the nozzle flow experiments of Katz.^[1] To date the comparison shows a discrepancy between the turbulence model computations and the experimental results. Much of the difficulty seems to lie with the turbulence model's ability to capture single phase separated flow. In addition work is progressing on examining three-dimensional effects due to a cross flow velocity at the inlet. The on-going work should be completed near the end of the summer.

REFERENCES

1. Gopalan, S. and Katz, J., "Flow Structure and Modeling Issues in the Closure Region of Attached Cavitation," *Physics Of Fluids*, Vol. 12, No. 4, April, 2000.
2. Chen, Y. and Heister, S.D., "Modeling Hydrodynamic Non-Equilibrium in Bubbly and Cavitating Flows," *J. Fluids Engineering*, V118, 172-178, 1995.
3. Nurick, W. H., "Orifice Cavitation and Its Effect on Spray Mixing," *Journal of Fluids Engineering*, Dec., pp. 681-687, 1976.
4. Henry, M. and Collicott, S.H., "Visualization of Cavitating Slot Flow," To Appear, *Atomization and Sprays*.
5. Xu, C., Blaisdell, R.A., and Heister, S.D., "Simulations of Turbulent, Cavitating Flows in an Injector Slot/Orifice," *Multiphase Flow 2001*, March 14-16, 2001, Orlando, FL.