

N94-24430

1993

NASA/ASEE SUMMER FACULTY FELLOWSHIP PROGRAM

MARSHALL SPACE FLIGHT CENTER
THE UNIVERSITY OF ALABAMA IN HUNTSVILLE

CFD SIMULATION OF COAXIAL INJECTORS

Prepared By: D. Brian Landrum, Ph.D.
Academic Rank: Assistant Professor
Institution and Department: University of Alabama in Huntsville
Department of Mechanical and Aerospace Engineering
MSFC Colleagues: Ten See Wang
P. Kevin Tucker
NASA/MSFC:
Offices: Structures and Dynamics Laboratory
Division: Aerophysics
Branch: Computational Fluid Dynamics

CFD SIMULATION OF COAXIAL INJECTORS

D. Brian Landrum, Ph.D.
Assistant Professor
Department of Mechanical and
Aerospace Engineering
University of Alabama in Huntsville

INTRODUCTION

The development of improved performance models for the Space Shuttle Main Engine (SSME) is an important, ongoing program at NASA MSFC. These models allow prediction of overall system performance, as well as analysis of run-time anomalies which might adversely affect engine performance or safety. Due to the complexity of the flow fields associated with the SSME, NASA has increasingly turned to Computational Fluid Dynamics (CFD) techniques as modeling tools.

An important component of the SSME system is the fuel preburner, which consists of a cylindrical chamber with a plate containing 264 coaxial injector elements at one end. A fuel rich mixture of gaseous hydrogen and liquid oxygen is injected and combusted in the chamber. This process preheats the hydrogen fuel before it enters the main combustion chamber, powers the hydrogen turbo-pump and provides a heat dump for nozzle cooling. Issues of interest include the temperature and pressure fields at the turbine inlet, and the thermal compatibility between the preburner chamber and injector plate. Performance anomalies can occur due to incomplete combustion, blocked injector ports, etc. The performance model should include the capability to simulate the effects of these anomalies.

The current approach to the numerical simulation of the SSME fuel preburner flow field is to use a global model based on the MSFC sponsored FDNS code (1). This code does not have the capabilities of modeling several aspects of the problem such as detailed modeling of the coaxial injectors. Therefore, an effort has been initiated to develop a detailed simulation of the preburner coaxial injectors and provide gas phase boundary conditions (species concentrations, pressures, temperatures, etc.) just downstream of the injector face as input to the FDNS code. This simulation should include three-dimensional geometric effects such as proximity of injectors to baffles and chamber walls and interaction between injectors.

This report describes an investigation into the numerical simulation of GH₂/LOX coaxial injectors. The following sections will discuss the physical aspects of injectors, the CFD code employed, and present preliminary results of a simulation of a single coaxial injector for which experimental data is available. It is hoped that this work will lay the foundation for the development of a unique and useful tool to support the SSME program.

PHYSICAL ASPECTS OF COAXIAL INJECTORS

Liquid propellant rocket injection is a complex combination of physical process including liquid atomization and evaporation, and chemical reactions. The complexity is increased by the fact that at least one of the constituents exists in both the liquid and vapor phases. In order to make the injection simulation problem numerically tractable, these physical processes are described by sub-models. The following two sections describe the sub-models for atomization and evaporation. The current study did not include the effects of chemical reactions and therefore this sub-model will not be discussed.

Injection / Atomization

In a coaxial injector the core liquid propellant jet is broken into smaller droplets through shear forces imposed by the co-flowing, high velocity, annular gas jet surrounding it. A cursory review of current atomization modeling capabilities and the experimental validation data base was recently presented by Liang, et al. (2). Currently, there are two primary approaches to the modeling of an atomizing liquid jet. The first approach, known as the Jet Embedding Technique (3), resolves the intact jet shape exactly with an adaptive grid. Simplified equations of motion are solved within the core to model its growth and subsequent atomization.

The second approach to atomization modeling is known as the Blob Atomization Model. This approach is based on the Reitz's approximation of the surface wave dispersion equation for a round jet (4) in conjunction with a Taylor Analogy Breakup model (5). The model assumes that the liquid jet can be represented by injected drops which are the diameter of the injection port. Linear stability theory is then used to model secondary breakup into smaller drops. Atomization is a function of droplet aerodynamics, liquid surface tension, and liquid viscosity. This approach does not allow the shape of the jet to be resolved. Numerically, the technique can be coupled to a Volume Of Fluid (VOF) technique (6), in which the fractional volumes of liquid, droplets and gas are tracked within each computational cell. The Blob Atomization and VOF approaches were used in the simulation described in this report.

Evaporation

A sub-model is also required to simulate the effects of evaporation of the cold liquid into the warmer surrounding gas. A vapor-liquid film model is used on the droplet surface. Quasi-steady state diffusion and energy equations are solved for the droplet heating rate and evaporation rate. The resultant equations used are presented by Liang and Ungewitter (see Reference 4).

For many injector scenarios the evaporation occurs at subcritical conditions where the droplet surface temperature is assumed to be the liquid saturation temperature. For the case of SSME preburner LOX injection, the chamber pressure far exceeds the critical pressure. In this situation the surface of the LOX droplet can be in a critical state while the interior of the droplet remains below the critical temperature. A supercritical evaporation model must ultimately be used. Reference 4 describes such a model although only subcritical evaporation was considered in the preliminary study documented in this paper.

COMPUTATIONAL CODE AND MODIFICATIONS

The numerical simulation was based on the Multiphase All-Speed Transient (MAST) code of Chen (7). This code uses a time accurate, temporal marching technique. The method is pressure based and also uses a operator-splitting algorithm to allow for various speed regimes in the flow field. A stochastic particle tracking method is incorporated (8). MAST uses a VOF technique, but simulation results indicate that this may not be totally active. The MAST code also includes a limited capability to generate computational grids. Options to generate uniform, exponentially stretched and mixed grids are available.

The MAST code was modified for this study. Although the numerical structure of the code is generalized for arbitrary fluid constituents, many thermofluid properties in the current version were hardwired for air. These properties had to be replaced with values representative of hydrogen and LOX. First, various thermofluid properties for the gaseous hydrogen were inserted. The second major task consisted of assembling a LOX data base. Required parameters included vapor pressure, latent heat of vaporization, surface tension, and viscosity of LOX as a function of temperature. A representation of the binary diffusion of oxygen into hydrogen also had to be provided.

EXPERIMENTAL DATA

A large experimental data base exists for coaxial injection using a variety of test liquids and gases. This data base is summarized in Reference 2. A capability for simulation of coaxial injection is currently being demonstrated at the Pennsylvania State University Propulsion Engineering Research Center (PSU/PERC). The hardware consists of a cylindrical chamber with an injector assembly at one end and a nozzle section at the other. The dimensions of the injector are comparable to the fuel preburner elements used in the SSME. Details of the injector assembly hardware are described by Pal, et al. (9). Both cold flow GN₂/H₂O and hot -fire GH₂/LOX injection has been performed in the laboratory to date. Because of the potential of this laboratory to produce validation data, a simulation of the PSU/PERC injector was chosen as the test case of this study.

INJECTOR SIMULATION RESULTS

The PSU/PERC chamber was modeled with an axisymmetric computational grid shown in Fig. 1. Only one-quarter of the length of the chamber was modeled. The upper half of the chamber was modeled so that the first grid line is the combustion chamber axis. For this preliminary investigation the numerically simulated injector did not include the LOX post recess. A fine uniform grid was used in the hydrogen annulus region. The grid was exponentially stretched from this region down to the chamber axis and upwards to the chamber wall. The total grid was 60 axial by 50 radial points. An injection boundary condition was applied at the hydrogen annulus and the downstream boundary condition was to fix the pressure at the quoted value for the hot-fire tests. The chamber axis was a symmetry boundary condition and all other surfaces were modeled with no-slip wall boundary conditions. Consistent with the blob injection used in the MAST code, LOX droplets were created at the $i=2, j=2$ grid point. These droplets could then convect or breakup in the chamber.

Several simulations were performed in order to investigate the capabilities of the MAST code. These consisted of hydrogen injection only, LOX droplet injection only and coaxial GH₂/LOX injection. Representative results are illustrated in Figs. 2 and 3 where the location of LOX droplet parcels in the computational domain are plotted at a time of 0.5 msec. Figure 2 shows the parcel distribution for LOX injection into static hydrogen. The droplets have penetrated a short distance into the chamber with no significant lateral dispersion. In Fig. 3 the LOX droplets are injected with the surrounding hydrogen jet. The axial penetration is comparable to the LOX injection only. The significant difference is the dispersion of the droplets laterally into the chamber. An interesting result of the simulation was that no droplet evaporation was seen during the time simulated. This may be due to the small magnitude of the temperature gradient between the LOX droplets (injected at 117 K) and the injected and ambient hydrogen gases (both at 289 K). This behavior may also indicate that the code is not accurately modeling the evaporation.

CONCLUSIONS / FUTURE WORK

A preliminary study of numerical simulation of GH₂/LOX coaxial injection has been performed. The MAST code was modified with thermofluid properties for hydrogen and oxygen. The modeled injector was based on hardware currently being used at Penn State University. Several aspects of the injection problem were simulated in order to evaluate the capabilities of the MAST code. Qualitative results indicate that the effects of the annular hydrogen jet are to disperse the LOX droplets laterally. No droplet evaporation was predicted. This may be due to the temperature gradients simulated or indicate a failure of the code evaporation model. Further analysis is required.

In general the MAST code was difficult to implement. Many of the thermofluid

parameters were hardwired for air and had to be changed. There is also some question as to whether the incorporated sub-models are correctly implemented. But, this criticism must be tempered by the fact that this is the first time that the code has been used to model a coaxial injection case. Further investigation into the code capabilities is therefore warranted.

Future work should include incorporation of H₂-O₂ gas chemistry into the simulation. The capability to model supercritical evaporation should also be included in the code. Detailed validation studies should then be performed using the Penn State GN₂/H₂O and GH₂/LOX data.

ACKNOWLEDGEMENTS

The author would like to acknowledge the technical assistance of Ten See Wang and Kevin Tucker during this project. Bruce Vu answered numerous questions about computer systems and plotting routines. Terry Jones provided word processing support. The contributions of each of these individuals was greatly appreciated.

REFERENCES

1. Chen, Y. S., "FDNS - A General Purpose CFD Code: User's Guide," *ESI-TR-93-01*, Engineering Sciences, Inc., May 1, 1993.
2. Liang, P. Y., Przekwas, A. J., and Santoro, R. J., "Propellant Injection and Atomization," Presented at the Combustion-Driven Flow Technology Team Meeting, NASA MSFC, July ??, 1993,.
3. Przekwas, A. J., Chuech, S., and Singhal, A. K., "Numerical Modeling of Primary Atomization of Liquid Jets," *AIAA 89-0163*, 1989.
4. Liang, P. Y. and Ungewitter, R., "Multi-Phase Simulations of Coaxial Injector Combustion," *AIAA 92-0345*, 1992.
5. Seung, S. P., Chen, C. P., and Chen, Y. S., "Development of an Atomization Methodology for Spray Combustion," presented at the 11th Workshop for CFD Applications in Rocket Propulsion, NASA MSFC, April 20-22, 1993.
6. Liang, P. Y. and Schuman, M. D., "Atomization Modeling in a Multiphase Flow Environment and Comparison with Experiments," *AIAA 90-1617*, 1990.
7. Chen, C. P., Jiang, Y., Kim, Y. M., and Shang, H. M., "A Computer Code for Multiphase All-Speed Transient Flows in Complex Geometries," NASA CR (unnumbered), October, 1991.
8. Kim, Y. M., Shang, H. M., Chen, C. P., and Ziebarth, J. P., "Numerical Modeling for Dilute and Dense Sprays," presented at the 10th Workshop for CFD Applications in Rocket Propulsion, NASA MSFC, April 28-30, 1992.
9. Pal, S., Moser, M. D., Ryan, H. M., Foust, M. J., and Santoro, R. J., "Flowfield Characteristics in a Liquid Propellant Rocket," *AIAA 93-1882*, 1993.

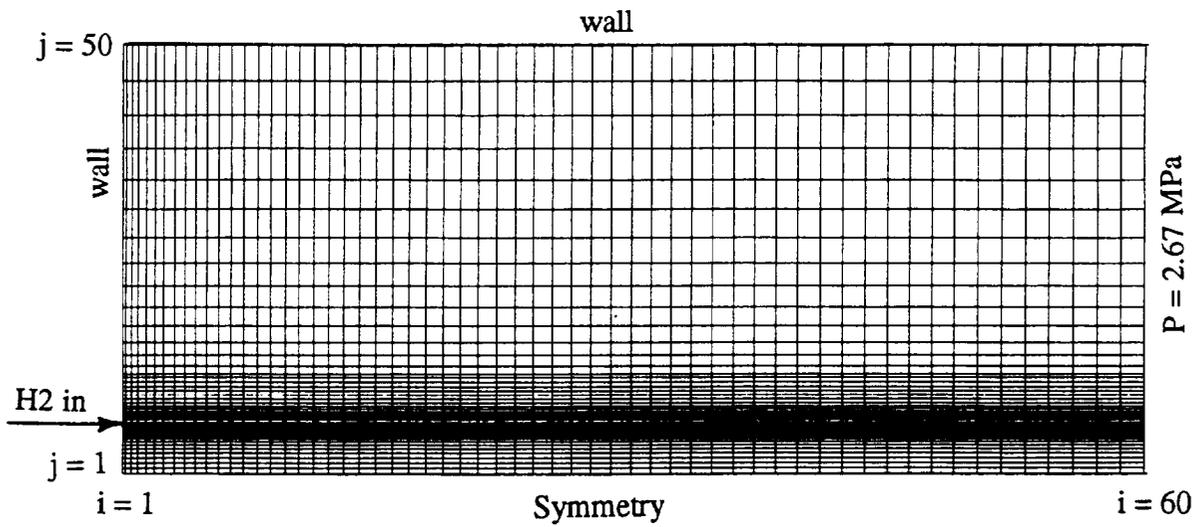


Fig. 1 Computational grid and boundary conditions for PSU injector simulation

Fig. 2 Spray parcel distribution for LOX injection only, $t = 0.5$ msec.

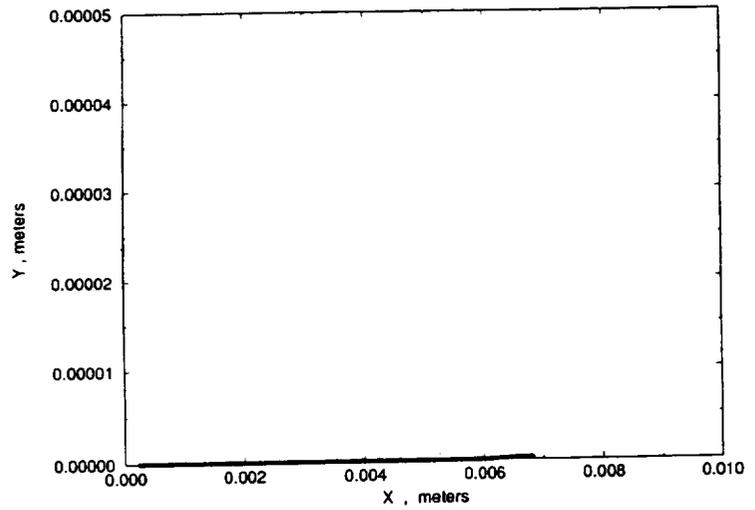


Fig. 3 Spray parcel distribution for GH2/LOX injection, $t = 0.5$ msec.

