TECHNICAL PROGRESS REPORT NO.16

INVESTIGATION OF HEAT TRANSFER AND COMBUSTION IN THE ADVANCED FLUIDIZED BED COMBUSTOR (FBC)

TO

U.S. DEPARTMENT OF ENERGY
FEDERAL ENERGY TECHNOLOGY CENTER
P.O. BOX 10940, MS 921-118
PITTSBURGH, PA 15236-0940

FOR

PROJECT NO.: DE-FG22-93MT93006

BY

DR. SEONG W. LEE, PRINCIPAL INVESTIGATOR

MORGAN STATE UNIVERSITY
SCHOOL OF ENGINEERING
BALTIMORE, MD 21239
(PHONE) 410-319-3137

OCTOBER 1997

ABSTRACT

This technical report summarizes the research conducted and progress achieved during the period from July 1, 1997 to September 30, 1997.

In order to conduct the numerical modeling/simulation on the advanced swirling fluidized bed combustor (hot model), the basic governing equations are formulated based upon the continuity and momentum equations, and energy equations in the cylindrical coordinates. The chemical reaction and radiation heat transfer were considered in this modeling/simulation work. The chemical reaction and the diffusion due to concentration gradients and thermal effects are also included in the modeling for simulation.

The flow system was configured in 3-D cylindrical coordinates with the uniform mesh grids. The calculation grid was set of orthogonal lines arranged in the cylindrical coordinates which includes three different directions: tangential direction (I), radial direction (i), and vertical direction (k). There are a total of 24192 grids in the system configuration including 14 slices of the tangential direction (I), 24 slices of the radial direction (j), and 72 slices of the vertical direction.

Numerical simulation on the advanced swirling fluidized bed combustor is being conducted using computational fluid dynamics (CFD) code, Fluent. This code is loaded onto the supercomputer, CRAY J916 system of Morgan State University.

Numerical modeling/simulation will be continued to determine the hot flow patterns, velocity profiles, static pressure profiles, and temperature profiles in the advanced swirling fluidized combustor.

TABLE OF CONTENTS

SECTION 1

NUMERICAL MODELING AND SIMULATION OF THE HOT COMBUSTOR MODEL

1.1 Introduction

Numerical modeling/simulation of gas-particle flows, heat transfer, and combustion process in the fluidized bed combustor have gradually increased with the development of modern computers. Numerical simulation has also been recognized as a powerful tool for design verification and operational guidance for the fluidized bed combustors [1,2). The successful simulation work may significantly reduce the efforts in the experimental study.

The cold flow patterns in the swirling combustor have been simulated by using the computational fluid dynamics code, Fluent (3], which were summarized in the previous reports [4].

The purpose of the numerical modeling/simulation on the advanced swirling fluidized bed combustor (hot model) is to determine the hot flow patterns, velocity profiles, static pressure profiles, species concentration profiles, and temperature profiles in the combustor chamber.

1.2 Basic Governing Equations

The basic governing equations for swirling, turbulent gas particle flows and combustion in the swirling fluidized bed combustor can be formulated based upon the continuity and momentum equations, and energy equation in the cylindrical coordinates. The continuity and three direction momentum equations [3,4] were introduced in the cold flow modeling/simulation [4].

Energy conservation equation [5] is as follows;

$$(h)/ t + \boldsymbol{(uh)/x} = \_(k*(T/x))$$

$$1/\mathbf{X}(sum)\ \boldsymbol{hJ} + u*(p/x) + T*(u/x) + S$$

Where T is the temperature, 1 is the heat flux of species, and k is the mixture thermal conductivity. Sh is a source term that includes sources of enthalpy due to chemical reaction (combustion reaction) and radiation heat exchange between the gas and the wall. The I,, is the static enthalpy which is defined as:

$$h=E\ mlhl$$

1.3 Species Conservation Equations

The conservation of species 1 is determined by:

$$1/t(M)+ 1/x(um)= 1/x(J)+S$$

Where m, is the mass fraction of species 1, ili is the diffusive mass flux of species 1 in the ith direction and S, is the net rate of production of species 1 per unit volume due to the chemical reaction.

In general, the diffusive mass flux, i,i is composed of diffusion due to thermal effects and diffusion due to species concentration and pressure gradients.

1.4 Radiation Heat Transfer Model

The chemical reaction and the radiation heat transfer were also considered in this modeling/simulation. The chemical reaction and the diffusion due to concentration gradients and thermal effects are included. Two radiation models are-available in CFD code, Fluent including the Discrete Transfer Radiation Model (DTRM) and the P-1 Radiation Model (3]. The DTRM for prediction of surface-to-surface radiation heat transfer with or without a participating medium were employed in our modeling/simulation.

In this model, the simplest case of a constant absorption coefficient is determined by the local concentrations of C02 and H20 species in the gas phase. The change in radiation intensity, DI along with a path ds is defined by:

$$dI/ds=-aI+ aoT^4/3.14$$

Where a = absorption coefficient (1/m)

a = Stefan-Boltzmann constant (W/m2.K 4)
T = gas temperature (K)

The radiation intensity approaching a point on a combustor wall surface is integrated to yield the incident radiation heat f lux grad as:

$$q(rad)= (1 -'E.)Id +ET$$

Where T,, is the surface temperature at a point P on the surface and e is the emissivity of combustor wall.

SECTION 2

FLOW SYSTEM CONFIGURATION FOR NUMERICAL SIMULATION

2.1 Flow System Configuration and Grid System

The swirling hot flow in the combustion chamber is a axially-symmetric 3-D turbulent flow problem involving chemical (combustion) reaction and radiation heat transfer. The system was configured in 3-D cylindrical coordinates with the uniform mesh grids. The computational cells and boundaries for the calculation domain are shown in Figure 1.

There are a total of 24192 grids in the system configuration including 14 slices of the tangential direction (I), 24 slices of the radial direction (J), and 72 slices of the vertical direction. Figure 2 shows the top view of computational domain in the combustor.

A typical slices in radial direction, I = 1 indicates the increased grid sizes in the radial direction as shown in Figure 3. it is seen in Figure 3 that a variable grid system with 1.5 increments of non-uniform spacings in tangential direction and 2 increments of non-uniform spacing in radial direction. This arrangement was to improve the overall accuracy of computation while still keeping the total number of grids, and the computer time and storage at reasonably low levels.

The combustor chamber has the height of 73 cm, the inner diameter of 14 cm, the outer diameter of 17.6 cm. Two sets of secondary air nozzles are installed on the combustor wall at two different levels of the height; 22.2 cm height for bottom nozzles and 41.6 cm height for top nozzles, respectively. Each nozzle is separated at 90 degrees around the combustor wall. The secondary air is injected into the combustor chamber from the nozzles with 45 degree of yam angle and zero degree of pitch angle.

A pressure gradient is expected to show in the across the cyclic planes at

I=O degree and I=90 degrees under the swirling flow in Figure 3. A initial mass flow rate across the boundaries will be provided and identified in the simulation process. The center line (or axis) of the combustor chamber is an axisymmetry line. Based upon the symmetry and cyclic boundaries, the simulation procedure will be simplified to save a lot of computer time.

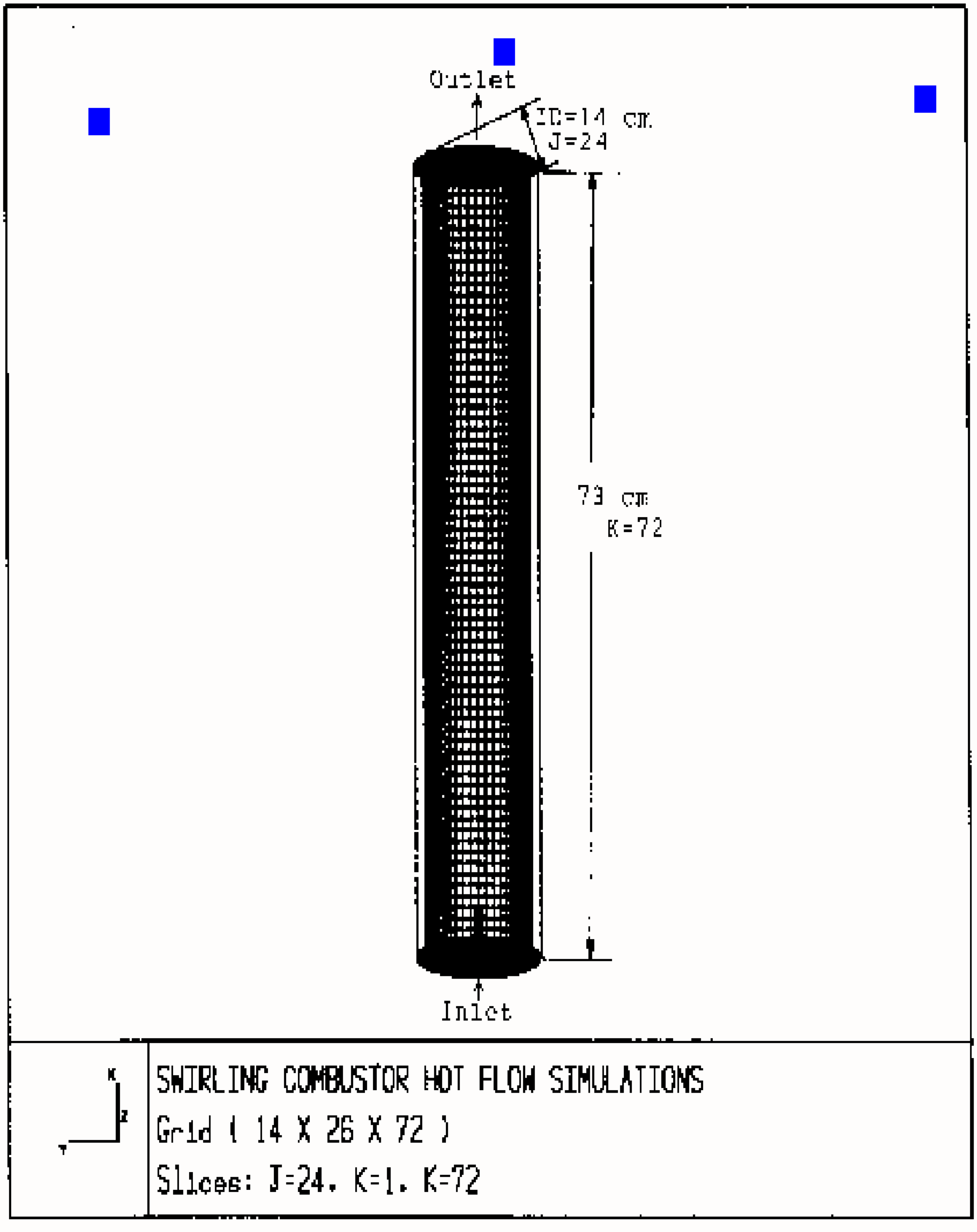

Figure 1 Flow System and Velocity Component in Combustor Chamber

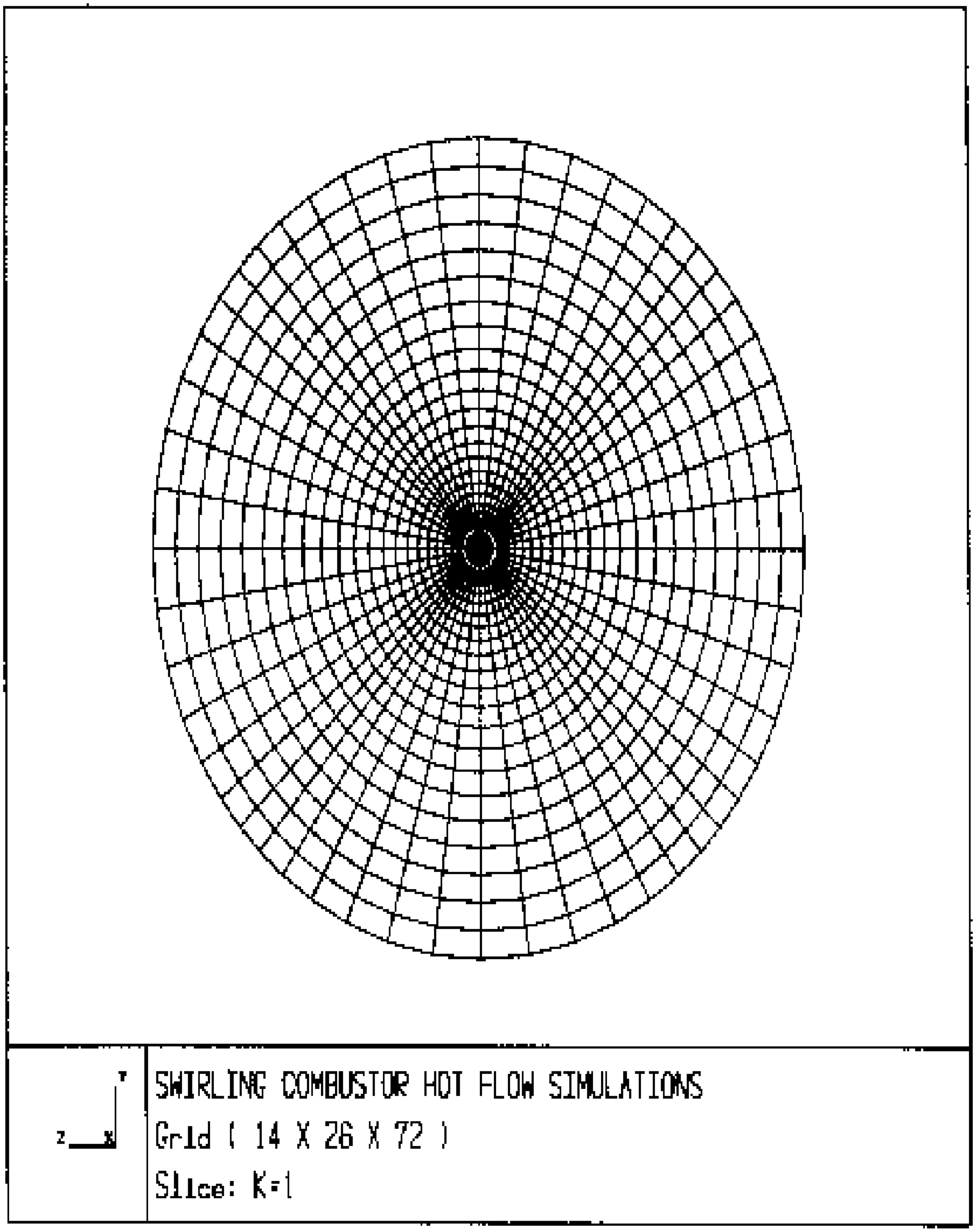

Figure 2 Top View of the Computational Domain in Combustor Chamber

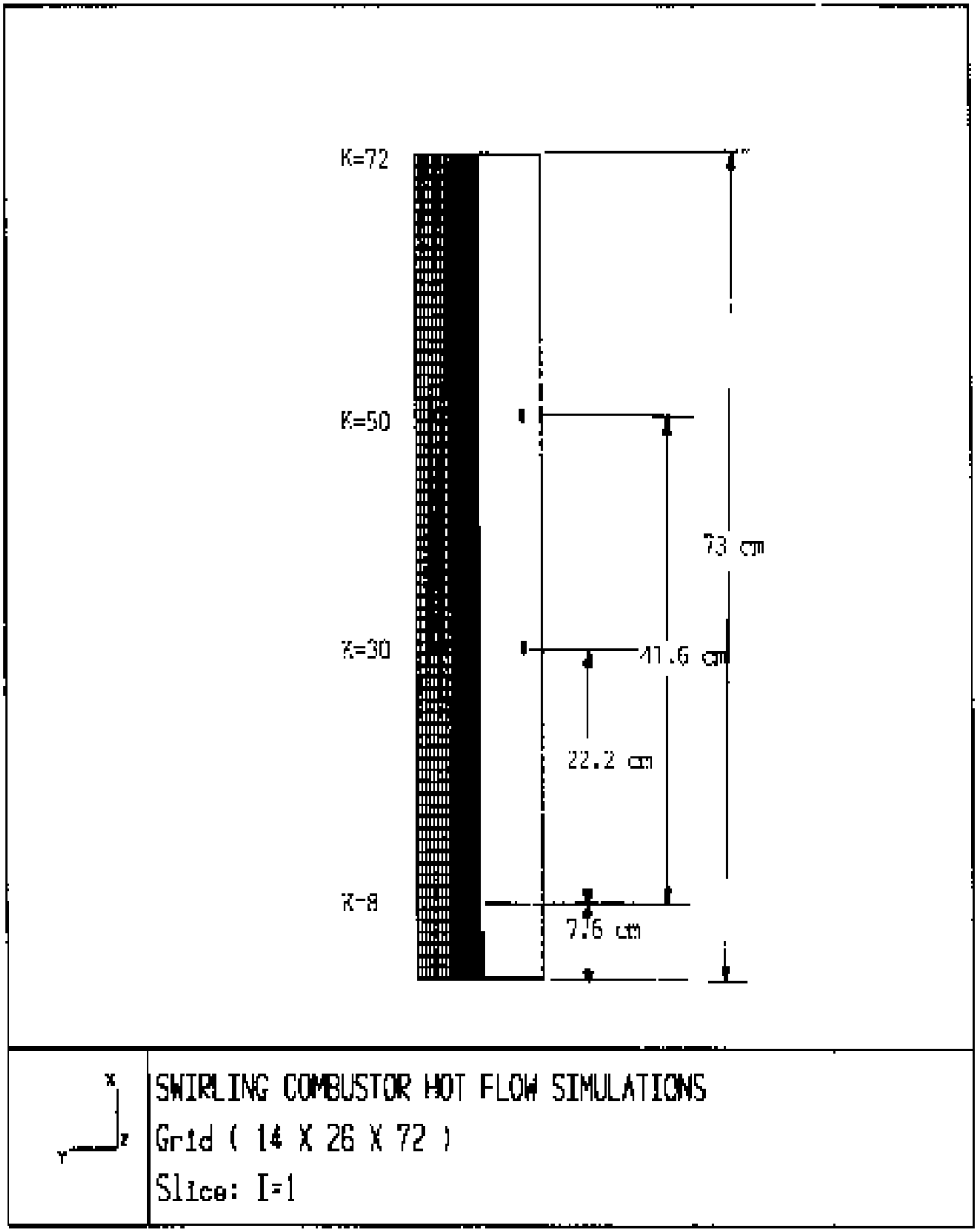

Figure 3 Computational Domain and Different Grid Spacings in Combustor Chamber

SECTION 3

CONCLUSIONS

The purpose of the numerical modeling/simulation on the advanced swirling fluidized bed combustor (hot model) is to determine the hot flow patterns, velocity profiles, static pressure profiles, species concentration profiles, and temperature profiles in the combustor chamber.

The basic governing equations are formulated based upon the continuity/momentum equations, and energy equations in the cylindrical coordinates. Species conservation equations are also described with the species concentration and pressure gradients. In addition, the chemical reaction and the diffusion gradients and thermal effect are included in the modeling for the simulation.

The flow system was configured in 3-D cylindrical coordinates with the uniform mesh grids. The calculation grid was set of orthogonal lines arranged in the cylindrical coordinates which includes three different directions: vertical direction (k), radial direction (J), and tangential direction (I).

The hot flow patterns in the advanced swirling fluidized bed combustor are being simulated by the computational fluid dynamics
code, Fluent.

The numerical simulation results will be discussed and presented with test conditions and input boundary conditions in the next report.

REFERENCES

(11 Boysan, F., et. al, Modeling Coal-Fired Cyclone Combustor, Combustion and Flame, Vol.63, 73-75, 1986.

[2] Khalil, E.E., Numerical Computations of Turbulent Flow Structure in a Cyclone Chamber, Joint ASME/AICHE 18th Natll Heat Transfer Conf., ASME Paper No. 79-Ht-31, 1979.

[3] Fluent User's Guide, Vol.4, Chapter 19, 1995.

[4] Lee, S.W. Technical Progress Report, No.9 and No. *10,* U.S. DOE, PETCII Jan. 1996 and April 1996.

[5] Hoffmann, K.A. et. al, Computational Fluid Dynamics, 3rd Ed. Vol.1; Chapt.9, Vol.2; Chapt.13, Engineering Education System, KS, 1995.