Control of Pollutant Emissions in Natural Gas Diffusion Flames by using cascade burners

SEMIANNUAL TECHNICAL PROGRESS REPORT

09/01/2000 through 2/28/2001

Dr. Ala Qubbaj, Engineering Department

03/30/2001

DE-FG26-00NT40831

University of Texas pan American
1201 West University Drive
Edinburg, Texas 78539-2999

DISCLAIMER NOTICE

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, or any of their employees, makes any warranty, express or implied, or assumes any legal liability or responsibility of 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 ant agency thereof. The views and opinions of authors expressed herein do not necessarily states or reflect those of the United States Government or ant agency thereof.

ABSTRACT

The advanced CFDRC software package was installed on a SUN-SPARC dual processor workstation (UTPA funded). The literature pertinent to the project was collected. The physical model was set and all parameters and variables were identified. Based on the physical model, the geometric modeling and grid generation processes were performed using the CFD-GEOM (Interactive Geometric Modeling and Grid Generation software). A total number of 11160 cells (248×45) were generated. The venturis in the cascade were modeled as two-dimensional axisymmetric convergent nozzles around the jet. With the cascade being added to the jet, the geometric complexity of the problem increased; which required multi-domain structured grid systems to be connected and matched on the boundaries. The natural gas/propane jet diffusion flame is being numerically analyzed. The numerical computations are being conducted using the *CFDRC-ACE+* (advanced computational environment) software package. The results are expected soon.

Table of Contents

EXPERIMENTAL (PHYSICAL) MODEL

The physical model of the proposed project is shown in Fig. 1, it includes in addition to the combustion chamber, flowmeters, a stainless steel burner of 2 mm internal diameter, natural gas cylinder, pressure regulator, as will as other tubing and connections. The combustion chamber is of 64 cm x 64 cm cross-section and 140 cm height. A circular opening is in the base plate covered with three layers of fine-mesh screens to allow uniform natural convection of air into the chamber. The top of the chamber will be connected to the atmosphere through an exhaust duct, as shown in Fig. 1.

The cascade consists of set four identical venturis which are arranged around the flame as shown in Fig.1. The venturis were modeled as thin wall solid boundaries. Four cascades (or sets of venturis) with different sizes were used. The cascaded flame is the baseline propane flame with the cascade of venturis installed around it. The cascade consisted of four identical venturis arranged at a certain spacing configuration.

NUMERICAL MODELING AND COMPUTATIONS

The facilities and equipment needed to perform the project are the (CFDRC) software in conjunction with a high speed SUN-SPARK Unix-based workstation. The software CFDRC along with a SUN-SPARC dual processor workstation were provided by the Engineering Department at UTPA.

A. Geometric Modeling and Grid Generation

The computational domain encompassed half of the flame jet (assuming axisymmetric flow conditions), extended to 64 cm in the axial direction and 14 cm in the radial direction (Fig. 2). A total number of 11160 cells (248×45) were generated with increasing spacing in the radial and axial directions; this provided adequate resolution where gradients were large, near the centerline, and saved CPU time where gradients were small, near the edges (Fig. 3). All important factors such as a right-handed grid, smooth transition from small to large cells, and grid orthogonality were taken into account in the grid generation process, and they did have a beneficial effect on the convergence of the solution. Moreover, the computational results were checked for grid independence.

The venturis were modeled as two-dimensional axisymmetric convergent nozzles around the jet. Four identical nozzles of the dimensions and spacing provided in Table (1) were used to form the cascade (Fig. 2). With the cascade being added to the jet, the geometric complexity of the problem increased; which required multi-domain structured grid systems to be connected and matched on the boundaries. The CFD-GEOM module (Interactive Geometric Modeling and Grid Generation Software) in the CFD-ACE+ package was used for geometric modeling and grid generation purposes. The natural gas/propane jet diffusion flame is being numerically analyzed. The numerical computations are being conducted using the *CFDRC-ACE+* (advanced computational environment) software package.

B. Reaction Model

The reaction model set was the instantaneous chemistry model in which the reactants are assumed to react completely upon contact. The reaction rate is infinitely rapid and one reaction step is assumed. Two reactants, which are commonly referred to as "fuel" and "oxidizer", are involved. A surface "flame sheet" separates the two reactants. The products are CO_2, H_2O, CO, NO , and OH.

The mass fractions for this model are computed by first using Eq. 1 to obtain the composition that would occur without the reaction. The "unreacted" composition, denoted by the superscript "u", is given by

$$(Y_i)^u = \sum_{k=1}^{K} \xi_{ik} f_k \quad \text{.....(1)}$$

where ξ_{ik} is the mass fraction of the i^{th} species in the k^{th} mixture, Y_i is the mass fraction of the i^{th} species and f_k is the mixture fraction of the k^{th} mixture. The change in composition due to the instantaneous reaction is then added to the unreacted mass fractions, as described below.

A stoichiometrically correct reaction step needs to be specified. The mass of species i produced per unit mass of fuel consumed is

$$r_i = -\frac{\nu_i M_i}{\nu_f M_f} \quad \text{.....(2)}$$

where ν is the stoichiometric coefficient of the species in the overall reaction; positive for product species and negative for fuel and oxidizer. The instantaneous reaction mechanism consumes either all the fuel or all the oxidizer, whichever is limiting. The amount of fuel consumed is

$$\Delta Y_f = \min\left(\frac{(Y_f)^u}{-r_f}, \frac{(Y_{ox})^u}{-r_{ox}}\right) \quad \text{.....(3)}$$

The change in each species due to the reaction is proportional to the change in fuel, with the proportionality constant given by Eq. 2. The mass fraction of each species is then given by

$$Y_i = (Y_i)^u + r_i \Delta Y_f \quad \text{.....(4)}$$

The right-hand side of the above equation is only a function of the K mixture fractions. Therefore, K-1 transport equations must be solved for the mixture fractions. These equations have no source terms due to chemical reactions.

C. Assumptions and Boundary Conditions

The following is a summary of the approximations and assumptions that have been made to simplify the numerical analysis:

1. Laminar, steady, and axisymmetric 2- dimensional flow
2. Newtonian fluid and ideal gas behavior
3. Uniform inlet velocity profiles
4. One-step, surface-sheet and instantaneous reaction
5. Soot-free diffusion flames
6. Negligible radiation losses from the flame

The geometry, coordinate system, and boundary conditions of the computational model are shown in Fig. 2. The boundary conditions were provided from the physics of the problem. Due to the symmetry around the x-axis, only one half of the flame is modeled. The experimental measurements indicated that the maximum value of half of the flame width, determined from the temperature profiles at x = 64 cm for the baseline case, is 7 cm. Therefore, both the centerline and the boundary at radius (y) = 14 cm are specified as symmetry lines in both the baseline and cascaded cases. The fuel inlet velocity is 11 m/s; based on the jet-exit diameter and Reynolds number (Table 1). The

venturis were simulated as thin-wall solid boundaries; by default this provides the no-slip condition (U=0, V=0) on both sides. A similar thin-wall boundary condition is also assigned for the tube-burner wall. The exit boundary is assigned a constant pressure (atmospheric) value.

RESULTS (pending)

The physical model was set, the geometric modeling and the grid generation processes were performed for the baseline and cascaded flames. Therefore, the numerical computations have just started. The results are expected soon.

CONCLUSION (pending)

As soon as the data become available, the conclusions will be made.

REFERENCES

Not available

TABLE 1

Conditions Used in the Computations

Fuel	Natural gas
Jet diameter	2 mm
Jet exit velocity	11 m/s
Convected air velocity	0.25 m/s
Venturi throat diameter, D	70 mm
Venturi height, H	70 mm
Venturi inlet diameter, DI	140 mm
Venturi locations*	x/d=0, 80, 160, 240
Axial locations*	near-burner (x/d=20)
	mid-flame (x/d=100)
	far-burner (x/d=180)
Ambient temperature	295 K
Ambient pressure	100 kPa

* See Fig. 2

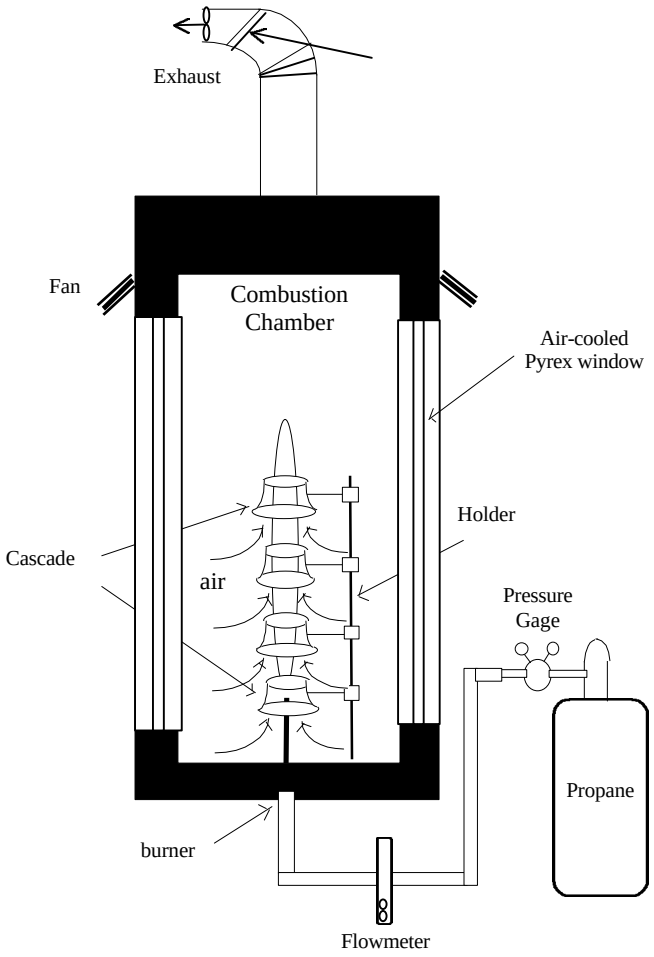

Fig. 1: The experimental (physical) model

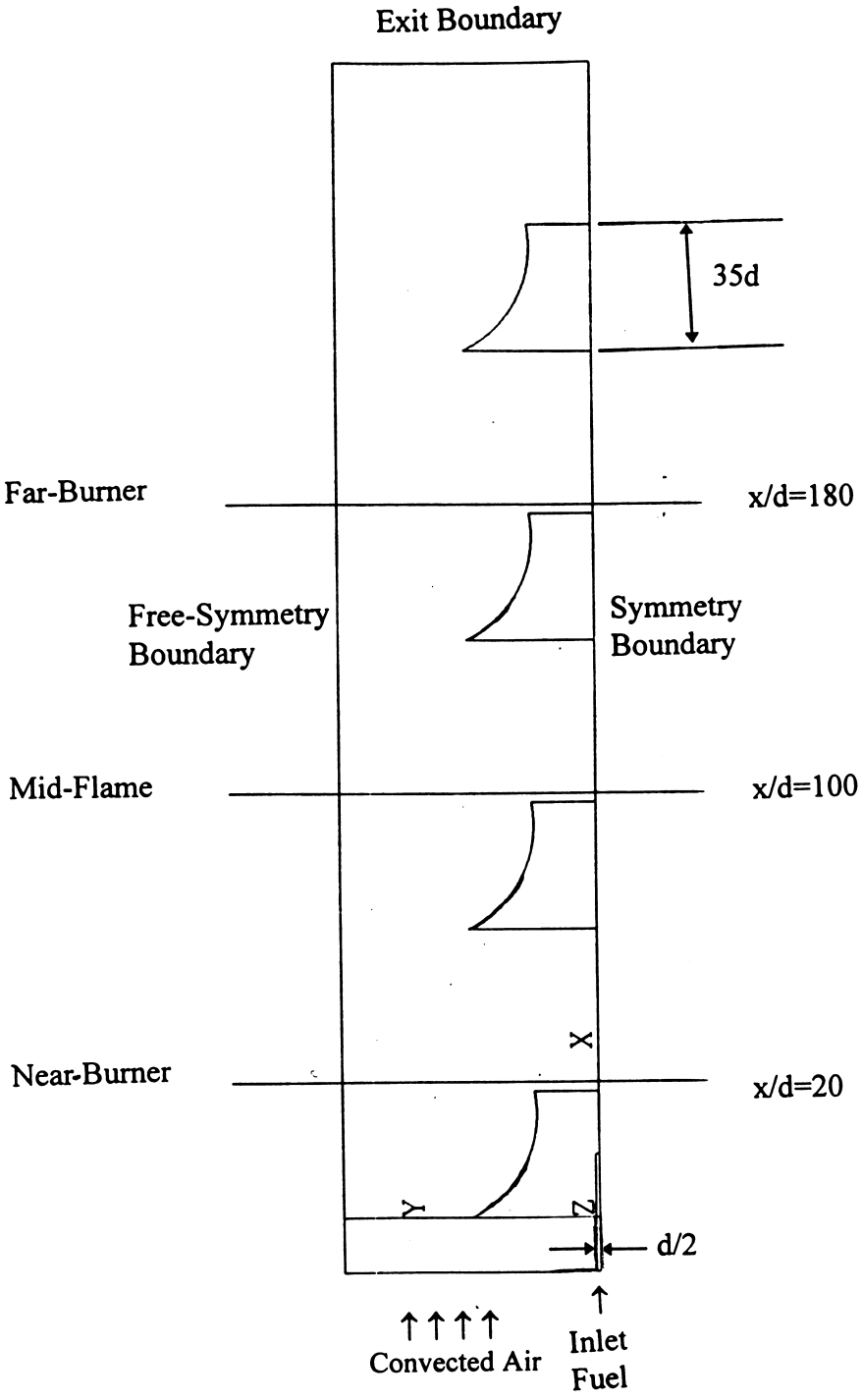

Fig. 2: Geometry and boundary conditions

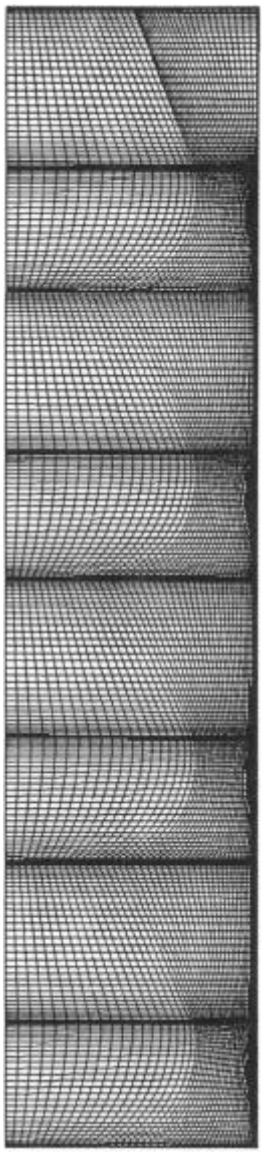

Fig. 3: Non-uniform grid generated for flow computations