Proceedings of 12th Asian Congress of Fluid Mechanics (12ACFM)

ASIAN CONGRESS OF FLUID MECHANICS

17 ~ 21 August, 2008
Daejeon Convention Center
Daejeon, Korea

Click Here

12ACFM Organization

12ACFM Local Organizing Committee

Chairman : **Hyung Jin Sung** (Korea Advanced Institute of Science and Technology, KAIST)

Co - Chairmen : **Seung Kyu Hong** (Agency for Defense Development)
Kyung Chun Kim (Pusan National University)
Nahmkeon Hur (Sogang University)
Dae Sung Lee (Korea Aerospace Research Institute)
Sang Joon Lee (Pohang University of Science and Technology)

Secretary General: **Deog Hee Doh** (Korea Maritime University)

Members :

Haecheon Choi
(Seoul National University)
Akinori Furukawa
(Kyushu University)
Choon-Man Jang
(Korea Institute of Construction Tech.)
Won Gu Joo
(Yonsei University)
Chongam Kim
(Seoul National University)
Cheolwan Kim
(Korea Aerospace Research Institute)
Heuy Dong Kim
(Andong National University)
Hyun Jung Kim
(Ajou University)
Jin Hyung Kim
(Small Business Corporation)
Jae-Soo Kim
(Chosun University)
Sa Ryang Kim
(Kangnung National University)
Han Seo Ko
(Sungkyunkwan University)
Oh Joon Kwon
(KAIST)
Inwon Lee
(Pusan National University)
Jeekeun Lee
(Chonbuk National University)
Seungsoo Lee
(Inha University)
Yeon-Won Lee
(Pukyong National University)
Hongwei Ma
(Beijing University)
Bock Choon Pak
(Chonbuk National University)
Jun Sang Park
(Halla University)
Michio Sadatomi
(Kumamoto University)
Toshiaki Setoguchi
(Saga University)
Jennifer Hyunjong Shin
(KAIST)
Chang Hyun Sohn
(Kyungpook National University)
Dong Joo Song
(Yeungnam University)
Simon Song
(Hanyang University)
Yong Kweon Suh
(Dong-A University)
Jaeyong Sung
(Seoul National University of Technology)
Baoguo Wang
(Beijing Institute of Technology)
Joon Yong Yoon
(Hanyang University)

Proceedings of the 12th Asian Congress of Fluid Mechanics
18-21 August 2008, Daejeon, Korea

NUMERICAL COMBUSTION MODELING OF A GAS-BURNER AND STUDYING ITS EFFECTING PARAMETERS

A. Kianifar [*1], N. Ghafoorianfar [*2], H. Moin [*3], I.R. Toroghi [*4] and M. Javadi [*5]

[*1] Department of Mechanical Engineering, Ferdowsi University of Mashhad, Iran, a_kianifar@yahoo.com
[*2] Department of Mechanical Engineering, Ferdowsi University of Mashhad, Iran, n.ghafoorianfar@yahoo.com
[*3] Department of Mechanical Engineering, Ferdowsi University of Mashhad, Iran, h_moin@yahoo.com
[*4] Department of Mechanical Engineering, Ferdowsi University of Mashhad, Iran, imrash84@yahoo.com
[*5] Department of Mechanical Engineering, Ferdowsi University of Mashhad, Iran, phd_member@yahoo.com

ABSTRACT: *In this paper numerical simulation of combustion over a sample of a prevalent gas-burner, and effects of parameters such as environment geometry, main parameters of chimney, effects of free and forced convection on the environment, in radiation modeling situation and without radiation has been studied. For study of radiation effects, a geometric model considering radiative heat transfer and without radiation is simulated and the results compared together. Furthermore, the distribution of environmental temperature in normal conditions, and advert to mass fraction of combustion products are the other cases which have been studied.*

1. INTRODUCTION

Simulation of combustion process with the purpose of studying the amount of pollutants produced by combustion needs perfect identification of this phenomenon from chemical point of view (hydrocarbon oxidization, quick reaction parameters, etc.), thermodynamic point of view (hypothesis of chemical reaction with infinity speed and computing the released heat), and fluid mechanics point of view (turbulent flow with change of density and have potential to produce turbulence in the case of releasing heat).

The best case for inflammation complex of natural gas is to observe fuel-air ratio 1 to 10. [1]. But if there is not enough air available, or the flame is not complete and uniform, there is not enough time for carbon monoxide to oxidize and convert to carbon dioxide and will be released.

2. MODELS AND GOVERNING EQUATIONS

The flow regime in the combustion chamber under studied is turbulent with change in density of the chemical species, which is aroused from the combustion. The governing equations on this phenomenon are conservation of mass, momentum, transmission of species and energy in the cylinder coordinate system with the assumption of steady with respect to time. For modeling the terms aroused from turbulent assumption k-ε method, and for modeling the combustion flow and calculating transmission of species Eddy-Dissipation method has been used [2].

$$\frac{\partial}{\partial x_i}(\rho u_i m_{j'}) = -\frac{\partial}{\partial x_j} J_{j',j} + R_{j'} \tag{1}$$

$$\frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial P}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} \tag{2}$$

Conservation of energy equation in this process with chemical reaction (combustion) is

$$\frac{\partial}{\partial x_i}[u_i(\rho E + P)] = \frac{\partial}{\partial x_i}(\lambda_{eff}\frac{\partial T}{\partial x_i} - \sum_{j'} h_{j'} J_{j'} + u_j(\tau_{ij})_{eff}) + S_h \tag{3}$$

Where S_h is the source term aroused from the heat released from the chemical reaction. Effective heat conduction factor, which is heat conduction factor of fluid and turbulence effects on it, with using the RNG k-ε method calculated as below:

$$\lambda_{eff} = \alpha . C_p . \mu_{eff} \tag{4}$$

For calculating the turbulence effects on the properties of flow and calculating the effective heat conduction factor and effective viscosity two assistance equations (k&ε) has been utilized [3].

In this study combustion of methane-air assumed with two stage combustion mechanism as below [1]:

$$CH_4 + 1.5O_2 \rightarrow CO + H_2O \qquad (step1), (reaction1),$$
$$CO + 0.5O_2 \rightarrow CO_2 \qquad (step2), (reaction2).$$

On the basis of this mechanism, the products of methane oxidization are carbon monoxide and water vapor. In the next stage carbon dioxide formed from carbon monoxide oxidization. Because of complete oxidization of methane in dilute complexes, in combustion with excess air the equation of combustion is expressed as [1]:

$$CH_4 + \frac{2}{\phi}\left(O_2 + \frac{100-\gamma}{\gamma}N_2\right) \Rightarrow CO_2 + 2H_2O + 2\left(\frac{1-\phi}{\phi}\right)O_2 + \frac{2}{\phi}\frac{100-\gamma}{\gamma}N_2 \qquad (5)$$

In this equation ϕ is the ratio of the amount of stoichiometric air to the amount of actual air and γ is the oxygen percentage exists in the air, which is 22% in normal conditions.

3. NUMERICAL MODELING OF COMBUSTION IN GAS-BURNER

One of the most important parts in numerical solving is producing proper geometry of the under studied system which has the least errors in meshing [6]. General structure of gas-burner with chimney is as below:

1- Furnace and crossing chamber of hot gases produced by combustion to the chimney.

2- Outer casing of the gas-burner.

Model which is used is shown in figure 1&2 with its geometric dimensions. It should be pointed out that modeling sensitivity is on furnace and crossing chamber of hot gases. One of the other important steps in numerical solving is producing proper grid with needed precision in different parts. Since radiation plays the main rule in heat transfer of bodies with atmosphere, effect of radiation phenomenon of gas-burner to the atmosphere is studied [4].

To calculate the mean heat transfer coefficient on the casing of gas-burner contacted with the atmosphere, the gas-burner is placed in cubic atmosphere with 3 meter length of each side which this atmosphere is contacted to 300K air from four sides. In figure 3 meshing model and whole view of the gas-burner and its atmosphere are shown. For modeling radiation the Discrete Ordinate model is chosen [5].

4. BOUNDARY CONDITIONS

In this modeling, entrance surface of fuel and air is considered to be a part of a cylindrical tube with a determined cross section. On the other hand, with determining the amount of fuel and air and mass fraction of each of them in entrance (with the assumption of combustion with excess air), velocity of entrance flow is calculated, and is given as entrance values. In this paper gas-burner is studied in two different ways:

1- Modeling of gas-burner and its surrounding in order to study atmosphere effects on efficiency of the gas-burner (e.g. room dimensions, distance between walls, and quota of radiation heat transfer to free convection)

2- Modeling of gas-burner and imposing atmosphere effects on gas-burner walls with imposing convection heat transfer boundary conditions (for studying distribution of different variables, manner of function and recognition of combustion phenomenon gas-burner, study the effect of chimney, etc.)

5. CONCLUSIONS

The results show that with 20 percent increase in gas-burner height, the amount of heat transfer will increase 18.5 percent, which is shown in figure 4. Also 20 percent increase in gas-burner width (from 250mm to 300mm) leads to 6 percent increase in the amount of heat transfer. In both situations, the reason of increase in heat transfer is raise in the exchange surface of heat transfer. Increasing lateral surface can increase the gas-burner efficiency saliently, but we can't change the gas-burner dimensions excessively because it has an elegant appearance and size and this is an important factor in residential areas. But we can create this surface increase with setting some fins on the gas-burner surface.

As the results of figure 5 shows, in the case of without radiation, temperature region of more than 310 K (those heated more than 5 degrees) is situated near the roof, and there is not considerable change in the temperature of beneath surfaces, which is the place for people presence.

We can note increase in radiation heat transfer contribution in heating equipments as one of the useful methods in reduction of energy consumption. In this method of heating, in equipments such as gas-burner, due to rather high temperature of its wall with respect to atmosphere, has some contribution in warming the atmosphere. As it is observed in figure 6 (in comparison with figure 5), with absorbing radiation energy from the gas-burner surface via ground, about 5°C (or K) temperature difference is created in height of 20 centimeters, which is increased with approaching the surface. This difference can be realized in figure 7, where temperature distribution produced by gas-burner with respect to height, in the center of the room, in two cases of with and without radiation effects is shown. It should be noted that the ground and the roof of the atmosphere is considered to be adiabatic in this paper and its heat dissipation is not took into consideration.

Another appropriate use of this heating is in small and close rooms without exiting of air flow from the environment. In this case we can achieve 15 percent efficiency increase. On the other hand, in small regions and with considering the convection phenomenon, radiation contribution in correct temperature and heat distribution is very high. Approximate 15 percent temperature increase in the room model with radiation is a proof of this claim, which can be seen from figures 5&6.

Modeling of this heating device in two geometric environments, which are room with four walls, and room with three walls and one open surface, which are the most usual uses of gas-burner, discloses advantages and disadvantages, temperature distribution and heat transfer trend in each of these environments. Nondimensional temperature distribution is a massive help to compare the gas-burner efficiency in these two environments. This comparison is shown in figure 8.

6. REFERENCES

[1] Subramaniam, S. and Howarth, D.C., "A PDF method for turbulent mixing and combustion on threedimensional unstructured deforming meshes," Int. J. Engine Res. 1: 171-190 (2000).

[2] Muradoglu, M., Pope, S.B., Caughey, D.A., "The hybrid method for the PDF equation of turbulent reacting flows: consistency conditions and correction algorithms," J. Comput. Phys. 172: 841-878 (2001)

[3] Adams, B. R., 1993a, "Computational Evaluation Of Mechanisms Affecting Radiation In Gas- And Coal-Fired Industrial Furnaces," Ph.D. Dissertation, University of Utah, Salt Lake City UT, USA

[4] "Fluent user's guide", Fluent Inc., 2001

[5] Tannehill, Anderson, Pletcher, Computational Fluid Mechanics and Heat Transfer, 2nd Ed., Taylor & Francis,Philadelphia, PA, 1997.

7. FIGURES AND TABLES:

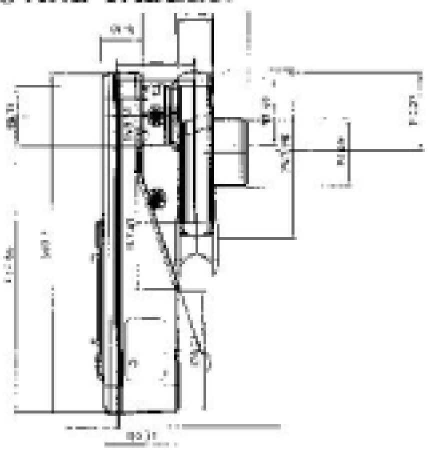

Figure1- Side view of the furnace of the gas-burner

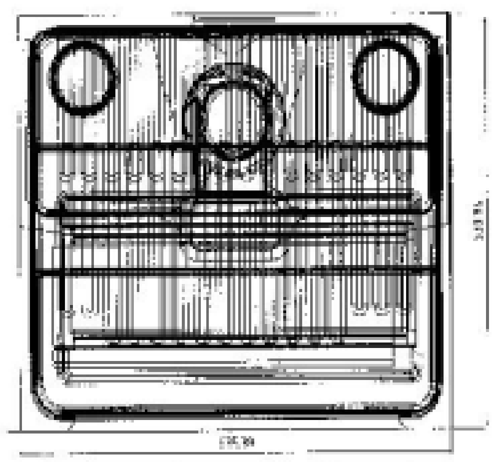

Figure 2- Front view of the furnace of the gas-burner

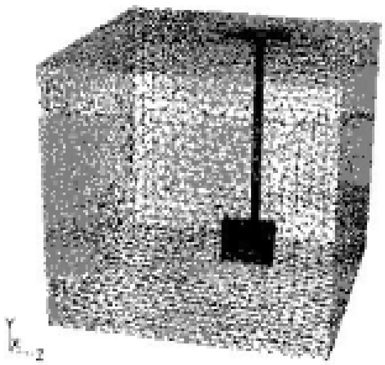

Figure3- Grids and overall view of the gas-burner and its environment

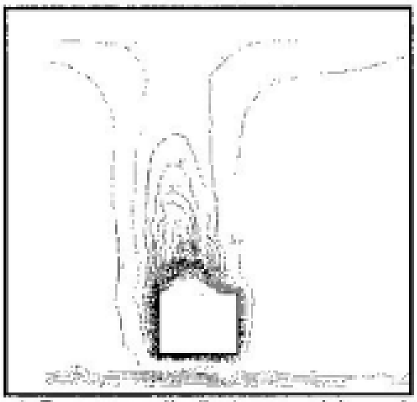

Figure6- Temperature distribution around the gas-burner with radiation effects

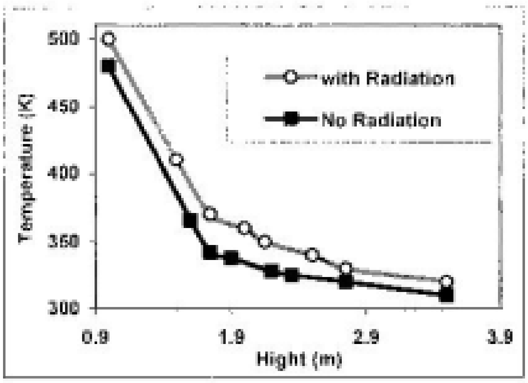

Figure4- Temperature distribution produced by gas-burner with respect to height, in the center of the room, in two cases of with and without radiation effects

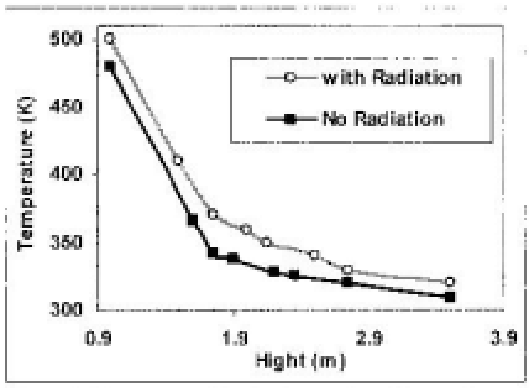

Figure7- Temperature distribution produced by gas-burner with respect to height, in the center of the room, in two cases of with and without radiation effects

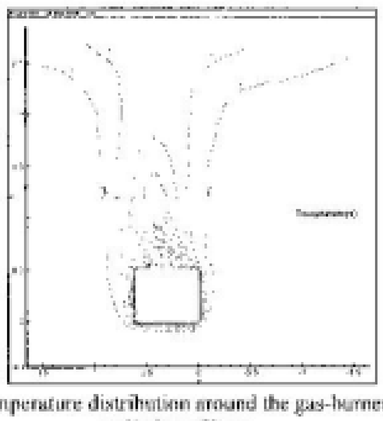

Figure5- Temperature distribution around the gas-burner without radiation effects

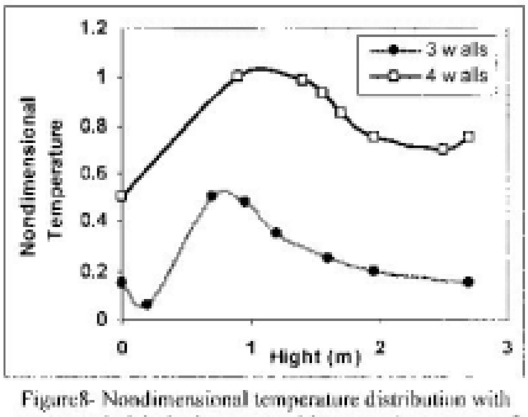

Figure8- Nondimensionall temperature distribution with respect to height in the center of the room, in two case of three and four walls