International Journal of Engineering Research And Advanced Technology (IJERAT)
ISSN: 2454-6135 **[Special Volume. 02 Issue.01, May-2016]**

Combustion and Heat Transfer Analysis of Gas fired heat exchanger unit using CFD

Vijaykumar G Tile, Pavan C S, Prajwal R, Shrinath K V, V Sachin Tendulkar
Assistant professor, Department of mechanical engineering,
MCE, Hassan

ABSTRACT

Understanding of combustion process and heat transfer rate and its performance is critical to complying with (progressively more) rigorous emissions requirements and to improving industrial productivity. Even small improvements in heat transfer rate will increase in energy efficiency and performance of the subsystems and which can have significant impacts in a continuous operation. While tremendous advances have been made in understanding the fundamental science of combustion process and the conjugate heat transfer, the remaining challenges are extremely complex. Today, trial and error is the only satisfactory method heat exchanger tubes design and problem-solving approach. To make improvements, it is critical to understand and accurately predict how heat is released and heat transferred across tube thickness and heat pickup by external air which is result in temperature rise by external air. Computational fluid dynamics (CFD) offers a numerical modeling methodology that helps in this understanding.

Combustible character of Natural gas (Methane) is one major area which uses Natural gas as a fuel. Hot combustible gas passes through inside the tube and cold air passes around the tubes understanding heat transfer across tube thickness and improve the design of tubes to increase the effectiveness of heat transfer is very important in design optimization of gas heaters system for HVAC industrial application. Burner and tubes are being the component where the combustion process takes place, produces emissions & heat transfer taken place across tubes thickness, thereby, making the fuels less environment friendly. But, with the use of natural gas as fuel for burner, the emissions are reduced to a considerable extent.

Keywords: *CFD, Tetrahedron Mesh elements, Heat transfer modes, Combustion, Combustor, Air to Fuel ratio, Producer Gas.*

INTRODUCTION

Conjugate heat transfer is an important process to be considered in designing the gas heat system to identify the hot and cold spots and calculate the temperature rise for external air which pass to the living room. It is well known that heat transfer study can be alleviated by modifying the flow field around the tube bundles. There are several techniques to the flow field that can maintain the uniform velocity distribution to develop effective heat transfer domain. The numerical results have been validated with literature data.

Combustion and heat transfer study is very essential in HVAC units on Gas heat subsystem which can optimize the energy utilization, thermal loads on gas heater tube wall and which will avoids the acids condensation at inside the tube.

Combustion is the process where the fuel and oxidant will react each other and release the energy for the designed rate of heat required to exchange the heat from hot fluid at inside the tube to exterior of the tube. It is important that understand the heat transfer phenomenon and effective burning inside the tube and heat pickup by tubes outer surface. Modeling of heat transfer through thickness is very complex work and least literatures are available on the same work. To optimize the gas heater subsystem for air heating is very important, effective utilization of energy and gas tubes wall temperature prediction. Wall temperature plays important role in terms of warranty periods of the subsystem. If the temperature attains higher than the thermal properties of the material which gets damage due to high thermal loads.

External air flow distribution on gas tube surface is also very significant to pick up the temperature over the tubes and control the temperature of hot and cold spots. Hot spots will damage the tube due to high temperature and cold spots will leads to condensate the sulphuric acids. These acids will condensate if the wall temperature lesser than the dew points, this leads to damage the tube through rust formation.

COMPUTATIONAL MODELING

Complete flow modeling of gas heat subsystem sections equipped with fully three-dimensional flow patterns. The gas heat wing model chosen for the present analysis is a HVAC (Heating Ventilating and Air Conditioning) system. This gas heat will be placed at downstream of the blower. When the blower pulls the low temperature from the ambient, the gas heat will be in operation mode. This gas heat will burns the designed amount of fuel inside tube section and it will help the low temperature air get increase the temperature through indirect heating method through conjugate heat transfer method. Assembly of gas heat subsystem as shown in the figure 1. Cylindrical shape with 1.2 mm thick tube is been using for burn the fuel and the tube and transfer the heat to exterior portion of the domain, tubers are in U-shaped, this type of arrangement will be helping the component assembly of arrangement the burners and the induced blower at same location. This arrangement will be easy to pull the ambient air for burning the fuel and to release the flue gas back to ambient.

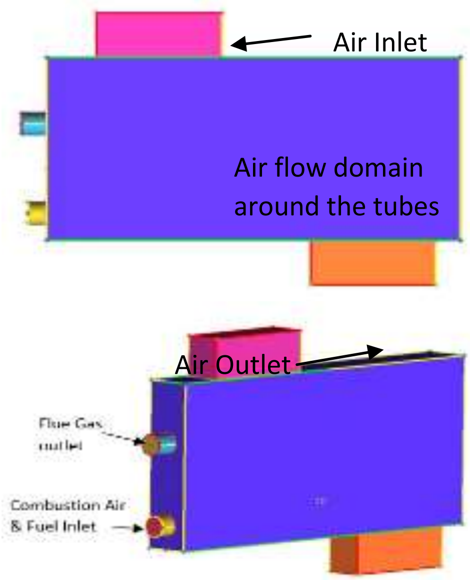


Figure1.Computational modeling of gas heat subsystem – Side, 3-D views

The geometrical and fluid domain details of the computational modeling that have been considered for studies are discussed in this section. Single row of gas tube is considered for CFD modeling to understand the flow behavior i) tube inside the combustion process and ii) heat transfer through the solid part of the tube finally iii) heat transfer to the exterior part of the tube in which air temperature will rise and pass through the living room. These three studies are more important to finalize the gas heat system in efficient way. This will also help the product development engineers to build efficient devices and reduce the cost of experimental process greater extent to finalize the design. CFD analysis will replaces the number of experiments for test build to validate the design parameters. Below are the computational modeling figures which includes tubes thickens considered for conjugate heat transfer study. It is necessary to have enough details of geometry of any test case for the purpose of modeling it on digital computer. Fluid domain is a virtually single tubes domain extracted for analysis purpose.The whole system whose outer boundaries are decided in such a way that the physics of the problem do not get affected. In all the present case studies which included different heat capacity of computational domain is the combustion will take place inside tube and air flow surrounding the geometry (external flow).

Figure2Computational modeling of gas heat tubes outer and inner walls

COMPUTATIONAL MESHED MODELS:

Tetrahedron meshing (unstructured) method was tried for tubes inner, thickness and exterior part of the compartments to get more connectivity to transfer the heat without and discontinuity. The mesh quality around thickness was generated little skewed elements due to manual arrangement of blocking approach. So tetrahedron elements are considered for CFD simulation the mesh quality was well within required criteria and mesh propagation was pretty well. Below are the figs about tetrahedron meshes.

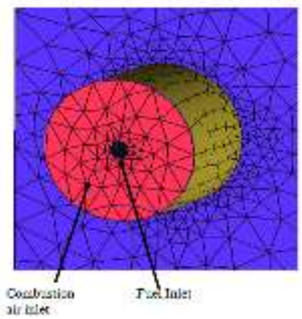


Figure3. 3-D view of Tetrahedron meshed model

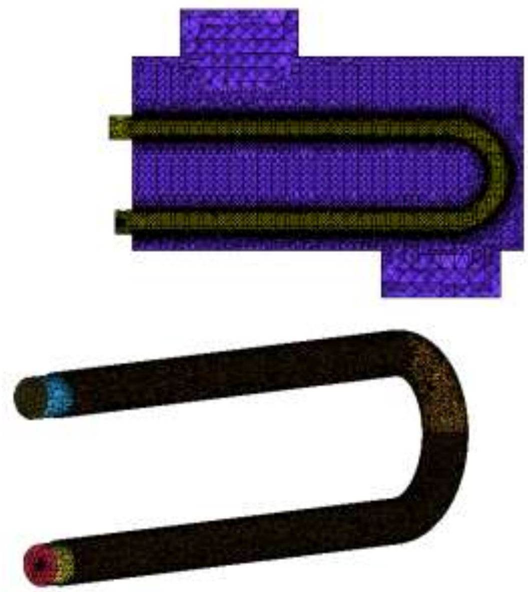


Figure4.Tetrahedron meshing model for tube inside and outside

RESULTS AND DISCUSSIONS FOR GAS HEAT CHAMBER

Simulation results have been generated for CFD model for tetrahedron meshed elements for study the combustion and heat transfer phenomenon in the gas heat domain. Natural gas using here as a fuel and ambient air is as oxidizer. Natural gas is passing through the center of the tube and ambient air is passing through around it, which can be seen in above figure. The aspect ratio for the gas to burn typically is 17 ratios, the 1 molecule of producer gas needed to burn 17 molecule of air. Below contours for various variables, velocity vectors and pressure plots gives insight of the combustion and heat transfer process phenomenon inside the gas heat domain.

Temperature contours plot inside the gas tube is generated. These contours are clipped range from 1073k to 2800k. High temperature is been captured at the entrance location where exactly gas and air intercept each other and then temperature reduces along the length of the tube. At the end one could see the temperature attained at 1000K, this temperature magnitudes are also observed in test. This simulation results are have good agreement with test.

PROBLEM STATEMENT:

Flow simulation was investigated in 3-D industrial standard code, CFX tools is extensively used for flow and thermal investigations. Inside tube combustion was performed, there are two inlets are considered for air and for one for fuel respectively. Air to fuel stoichiometry was maintained to completely burned indie the tube and tubes wall thickness also modelled in the CFD to investigate the conjugate heat transfer, due to this conjugate process heat will be transferring into the exterior domain. This will be further heat transfer take place through convection to pick-up the heat and this will finally release for room hot air purpose. External air targeted is velocity and blower outlet cross section, which is cold air and its mimic the realistic in nature in which air can flush around the tubes. The combustion simulation uses the standard RANS equations for mass, momentum and energy and the turbulence is simulated with the SST model known to have given the most acceptable results in the validation made in the earlier investigation. A no-slip and adiabatic wall conditions are imposed at the wall of the combustor domain. Outlet maintain the almost uniform temperature across the outlet and it can connect for any application like boiler, domestic purpose, etc., The numerical code uses an industry standard CFD code solving 3D RANS equations with a second order upwind scheme. The solution sought is for steady state and a diminishing residuals falling by 1×0^{-6} is set as the criteria for convergence. For a clarity the main equations considered for the solution is provided below.

1. The Continuity Equation:

$$\frac{\partial \bar{U}_j}{\partial x_j} = 0$$

2. The Momentum Equation:

$$\frac{\partial}{\partial t}(\rho\bar{U}_i) + \frac{\partial}{\partial x_j}(\rho\bar{U}_i\bar{U}_j) = -\frac{\partial \bar{P}}{\partial x_i} - \frac{\partial}{\partial x_j}\left(\bar{\tau}_{ij} + \rho\overline{u_i'' u_j''}\right)$$

3. The Energy Equation:

$$\frac{\partial}{\partial t}(\rho\bar{h}) + \frac{\partial}{\partial x_j}(\rho\bar{U}_j\bar{h}) = -\frac{\partial}{\partial x_j}\left(Q_j + \rho\overline{u_i'' h'}\right)$$

Results and Discussion

Simulation results have been generated for CFD model for tetrahedron meshed elements for study the combustion and heat transfer phenomenon in the gas heat domain. Natural gas using here as a fuel and ambient air is as oxidizer. Natural gas is passing through the center of the tube and ambient air is passing through around it, which can be seen in above figure. The aspect ratio for the gas to burn typically is 17 ratios, the 1 molecule of producer gas needed to burn 17 molecule of air. Below contours for various variables, velocity vectors and pressure plots gives insight of the combustion and heat transfer process phenomenon inside the gas heat domain.

Temperature contours plot inside the gas tube is generated. These contours are clipped range from 1073k to 2800k. High temperature is been captured at the entrance location where exactly gas and air intercept each other and then temperature reduces along the length of the tube. At the end one could see the temperature attained at 1000K, this temperature magnitudes are also observed in test. This simulation results are have good agreement with test.

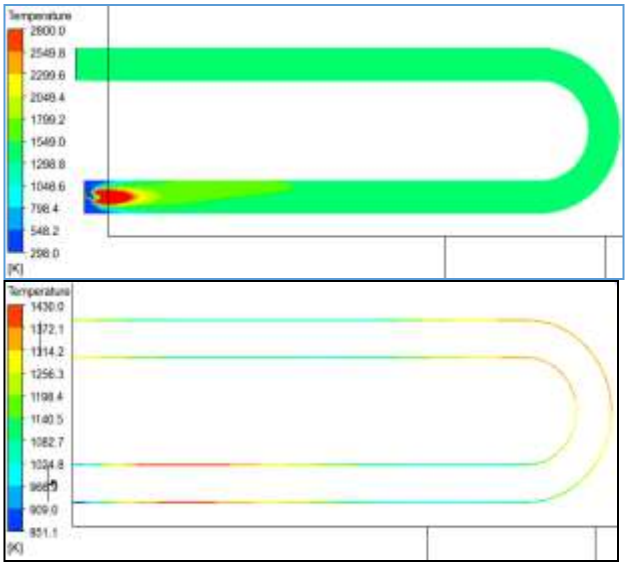


Figure5. Temperature contours inside the tubes and temperature contours in solid part of the steel.

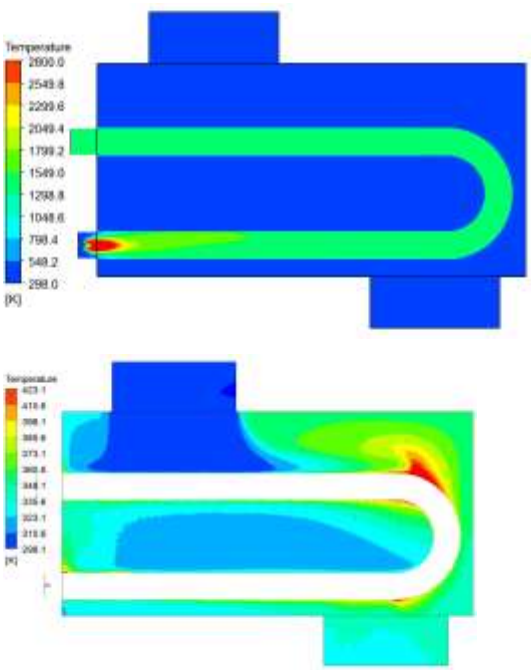


Figure6. Temperature contours inside the exterior portion of the gas heat system

Through these exercises is very much important to understand the temperature profile of tube wall, below figure shows the temperature contours on tube inner wall and outer wall. Inner wall 1578k at the bottom row of the tube and top row shows the 1250k range. The inner is wall of the tube will be exposed to burning face of fuel which is combustion. Outer tube wall will attain to 565 k to 310 k which is because the external air will flush and pick-up the heat from the tube. This thermal physics could be seen in below figures.

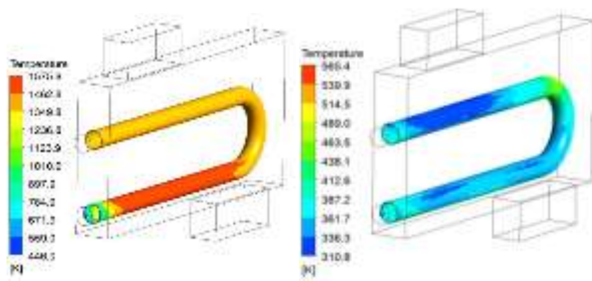


Figure7. Tube walls temperature i) inner wall ii) outer wall of the tube

Velocity vectors of exterior air flow and flue gases flow inside the tube can be seen in below figure. Exterior air velocity enters into the domain from blower exit and then passes over tubes then followed through the outlet duct. This flow path will flush the air around tubes and pick up the heat and hot fluid will pass through the outlet duct for leaving room applications.Fuel and ambient air enters through inside the tube and burn the fuel with reacting with oxidant and releases the products which are CO2 and H2O. These flue gases flow along tube length, then releases into the ambient. Below figure shows the velocity vectors for flue gas and exterior air flow.

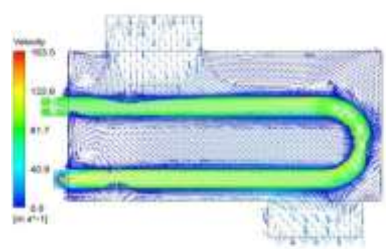
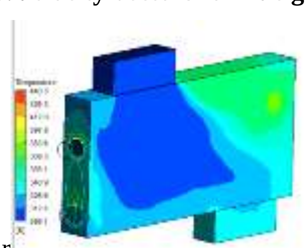


Figure8.Velocity vectors for flue gas and



air

Figure9.Temperature contours around duct

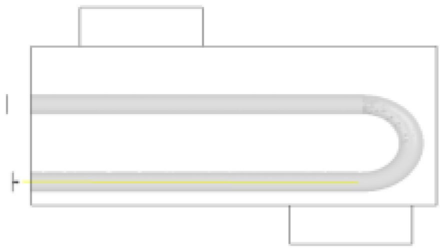
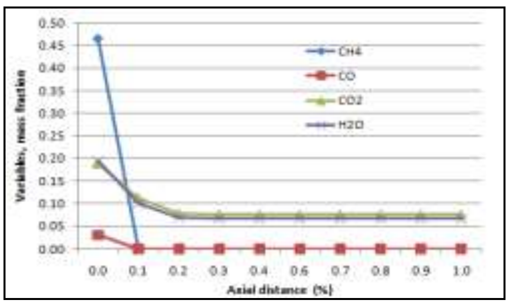


Figure10. Flue gas variables along bottom part of tube



. **Figure.11 Line selected for variables observations**

CH_4 is the energizing composition for natural gas below Fig.5line plot shows the methane behavior inside the tube. We can see that the methane and carbons are can burn inside the tube and which releases the energy which can transfer heat, which is burning the methane content at the entrance of tube. This plots can say that whole methane composition is effectively burned inside the tube. There are no unburned hydrocarbons releasing through gas tube outlet.

CONCLUSION:

Combustion and heat transfer study is very much required in HVAC units on Gas heat subsystem which can optimize the energy utilization, thermal loads on gas heater tube wall and which will avoids the acids condensation at inside the tube. Combustion is the process where the fuel and oxidant will react each other and release the energy for the designed rate of heat required to exchange the heat from hot fluid at inside the tube to exterior of the tube.

Flue gas variables can see that all the energy values like CH4 and CO are burning completely at bottom portion of the tube. And complete burn has taken place in first row of the tube and there is no unburned hydrocarbons are releasing.

REFERENCES

1. Rajagopal THUNDIL KARUPPA RAJ* and Srikanth GANNE;Shell side numerical analysis of a shell and tube heat exchanger considering the effects of baffle inclination angle on fluid flow: Thermal Science: Year 2012, Vol. 16, No. 4, pp. 1165-1174
2. J. Faison A. Davis D.B. Post; Created Lynn Haven Gas Heat Test Manual from ANSI Z21.47-2003 and Gas-Fired Central Furnace Test Procedure Manual (Revision H, January 14, 2000)
3. Pietro Asinari; Radiation Heat Transfer: Basic Physics and Engineering Modeling; Spring 2007, TOP – UIC Program University of Illinois at the 'Politecnico di Torino'
4. Hong Xu, ChokriGuetari, Kurt Svihla; Simulation of Free Convection with Conjugate Heat Transfer; ANSYS, Inc.
5. http://en.wikipedia.org/wiki/Combustor