International Journal of Emerging Engineering Research and Technology
Volume 3, Issue 9, September, 2015, PP 1-7
ISSN 2349-4395 (Print) & ISSN 2349-4409 (Online)

Design of a Small Scale CFB Boiler Combustion Chamber for Laboratory Purposes

Dr. Bivek Baral, Mr. Anup KC, Mr. Pratisthit Lal Shrestha, Avip Bastakoti, Anil Kumar Pachhain, Shreedhar Neupane

Department of Mechanical Engineering, Kathmandu University, Dhulikhel,Nepal

ABSTRACT

The biomass contains stored energy from the sun and is a renewable source of energy. Fluidized bed combustion (FBC) is one of the most promising energy conversion options available today for the solution of crisis of rapid energy usage. FBC combines the high efficiency combustion of low grade fuel e.g. rice husk, wood and other agricultural waste products. The rice husk/rice straw is one kind of renewable energy source which is abundant in agricultural country like Nepal. We have used modelling tools like Solid works for the design of a laboratory sized CFB boiler and the simulation software ANSYS FLUENT. We have carried out steady state combustion to verify the results. The main objective behind our study was the design of a CFB boiler for laboratory purposes. The methodology followed was based on literature survey, design of fuel hopper, combustion chamber and cyclone and finally using the scenario we created a simulation in ANSYS FLUENT. Finally, the parametric excel sheet was made specifically so that it can give the dimensions of a large variety of commercial boiler in varying environmental conditions.

Keywords: FBC, CFBC, fluidized bed, biomass combustion, low pollutants.

INTRODUCTION

Increase in energy consumption has led human beings to discover many kinds of energy conversion technologies. One of the promising and highly applicable is the Combustion Fluidized Bed (CFB) technology which has grown in popularity in the recent years. The main objective of this project is to enable the use of this technology in agricultural based country like Nepal where biomass waste could be used to power the boiler and obtain useful energy.

This paper mainly concerns with the design (thermal) of a circulating CFB boiler combustion chamber for laboratory purpose and a very minimalistic simulation process to give anyone about the combustion taking place inside the boiler. The use of CFB boiler has many advantages over other conventional methods of steam generation as there is reduced NO_x and SO_x emission including the use of low-grade fuel to obtain high power output.

In the field of CFB boiler there are two main principles: the bubbling bed and the circulating bed. The differentiation of bubbling bed and circulating bed can be done by calculating the core particle velocity. A bubbling bed usually has velocity less than 3 m/s whereas for the circulating bed it is higher than 3 m/s. However, this definition is completely satisfied in case of large industrial boilers. Another distinction is that the circulating bed has a cyclone assembly to recycle the fly ashes. [1]

Wide range of fuel size, moisture content, biomass content, sulphur reduction, constant bed temperature and most important high power output has led all kinds of research in the field of fluidized combustion technology.

DESIGN PROCESS

The design of the boiler is divided into 3 sub sections: fuel hopper design, combustion chamber design and the cyclone design.

Main purpose of feeding system i.e. fuel hopper is to continually feed fuel in the combustion chamber. For this purpose a screw conveyor as best suitable feeding system for our boiler. There are many other

Address for correspondence:

avipbastakoti007@yahoo.com

but as our study background it is easier for us to design. It can move solid particles from a low pressure zone to a high-pressure zone with a pressure seal. By varying the RPM of its drive, a screw feeder can easily control the feed rate.

The relation between the rice husk flow with the diameter, pitch, fillet height and revolutions of the screw, is given by the expression.

$$M = 60\pi\varphi Snr_h(D_sh_s - h_s^2) \tag{1}$$

And the fillet height, the outer diameter and the axis diameter, were related by the following expression:

$$d = D_s - 2h_s \tag{2}$$

The furnace of the boiler is designed by taking the standardized combustion process with excess air of 20%. The indirect method used for efficiency calculation is taken as a basis for the design of the required domain for the furnace. Specifying the resident time, using the stoichiometric relations and determining the environmental parameters the dimension of the furnace and minimum fluidization velocity were calculated. [2]

The equations involved in stoichiometric calculation are,

$$C + O_2 = CO_2 + 32,790\frac{kJ}{kg}\ of\ carbon$$

(3)

$$C_nH_m + \left(\frac{n+m}{4}\right)O_2 = nCO_2 + \frac{m}{2}H_2O + heat \tag{4}$$

$$S + O_2 = SO_2 + 9260\frac{KJ}{kg}\ sulphur \tag{5}$$

Although rice-husk fuel is very low in sulphur content, limestone in the mixture was added as a necessary precaution and also for additional mobility during the fluidization. [1]

For the cyclone, Stairmand High Efficiency (SHE) cyclone design was adopted as a reference because of its flexibility and higher efficiency and performance. [3]

In case of pressure vessels like boiler the actual measurement of any device is measured on the basis of equivalent evaporation which is defined as the rate in kg per hour at which water would be vaporized in standard condition and given by the following equation, [4]

$$m_e = m_a\frac{h_g - h_{f1}}{h_{gh}} \tag{6}$$

Computational fluid dynamics (CFD) is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations which govern these processes using a numerical process. [5]

There are set of equations that are solved inside ANSYS during the calculation in successive iterations. These are predefined equations in ANSYS library. Through the input boundary conditions by user, these equations are solved to get required result. [6]

For the non-premixed combustion model, ANSYS FLUENT solves the total enthalpy form of the energy equation.[5]

$$\frac{\partial}{\partial t}(\rho H) + \nabla(\rho v H) = \nabla\left(\frac{k_t}{c_p}\nabla H\right) + S_h \tag{7}$$

ANSYS FLUENT 15.0 is used in solving the mesh created by ICEM CFD 15.0. Non premixed combustion is taken into consideration for the rice-husk combustion in the combustion chamber. The coal calculator is used to input the ultimate analyses of rice-husk and setting the boundary conditions in the inlet and outlet the results were obtained. Standard initialization for the steady state combustion was done and the solution was computed from the inlet. Then the solution was initialized as convergence was obtained.

Results of the temperature, density, mass fraction of various species were obtained as a solution. The meshed file from ICEM-CFD was imported to ANSYS FLUENT and applying the boundary condition from the design, using the ultimate analyses from the rice-husk a non-premixed combustion was observed in the results. Energy, P1 radiation and discrete phase model are used. The turbulence models are the two-equation models are the standard K-ε model is used our simulation. It is widely used in turbulence state simulations because of its general applicability, robustness and economy. The two transport equations for the kinetic energy and dissipation rate are solved to form a characteristic scale for both velocity and length.

RESULTS AND DISCUSSION

The design of the furnace was the initial design step in the project and was carried out in MS Excel and as the design of the fuel hopper and cyclone were dependent on the design of furnace they were designed accordingly.

In case of fuel hopper, the revolution of dosing screw and feeding screw are 6 and 16 rpm respectively in our case and the volume of hopper is $0.8m^3$ and the feed rate of rice husk is 48.96 kg per hour.

In case of furnace, the required dimensions of the obtained are 0.84m X 0.42m (L X B) and height as 2.8m respectively and the minimum fluidization velocity as 0.7 m/s to start the fluidization process.

In case of cyclone separator, the height was obtained as 1m along with diameter of cyclone as 0.25m and gas outlet diameter of cyclone as 0.125m.

From above relation the calculated equivalent evaporation was 30.177kg steam /kg of fuel.

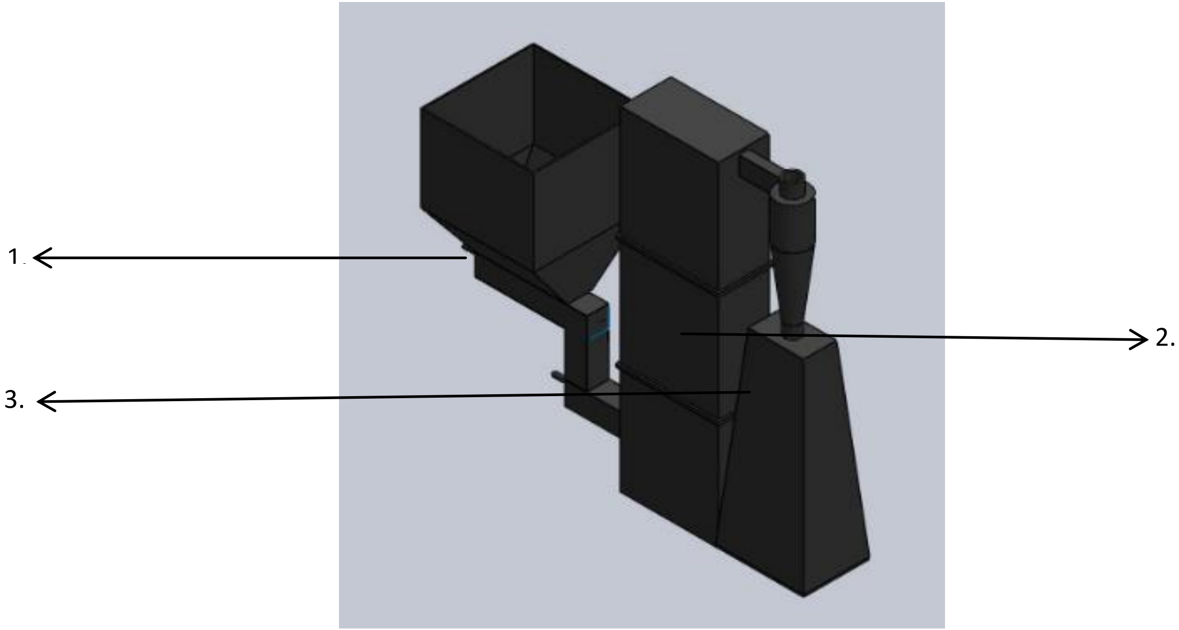

Figure1. *Final Design*

The above figure represents the final design of the complete CFB boiler combustion chamber (2) with hopper (1) and cyclone-syphon (3) assembly.

The results of the simulation tests in FLUENT 15.0 are given below:

Temperature Distribution

When the fuel and air enter into the combustor, it burns due to high velocity and temperature and then temperature increase rapidly in the combustor. Finally the result, the total temperature of the rice husk. The Figure 2 shows that the temperature profile in circulating fluidized bed combustor. In this figure, the bed and temperature are increasing as soon as the rice husk particles are burning and finally obtained the maximum value of temperature after coal combustion is 1869K.

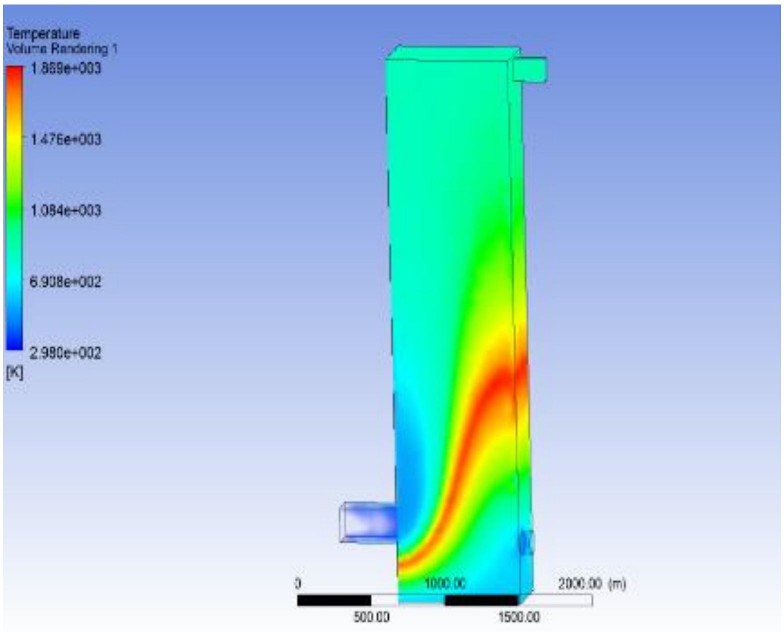

Figure2. *Contour plot of temperature*

CO and CO_2 Mass Fraction

According to given contour plot Figure 3, CO mass fraction is zero at the lower end of the furnace this is because as the fuel and air are mixed combustion could not occurred at the starting time after certain time combustion of the rice husk occurred at the upper part of chamber and maximum value was found 0.14.

Maximum value of the CO_2 faction is at the middle of the chamber because at that part complete combustion occurred. From Figure 4 we found that all rice husks could not burn at the furnace in first phase. But unburned fuel will burned when it is circulated from the cyclone.

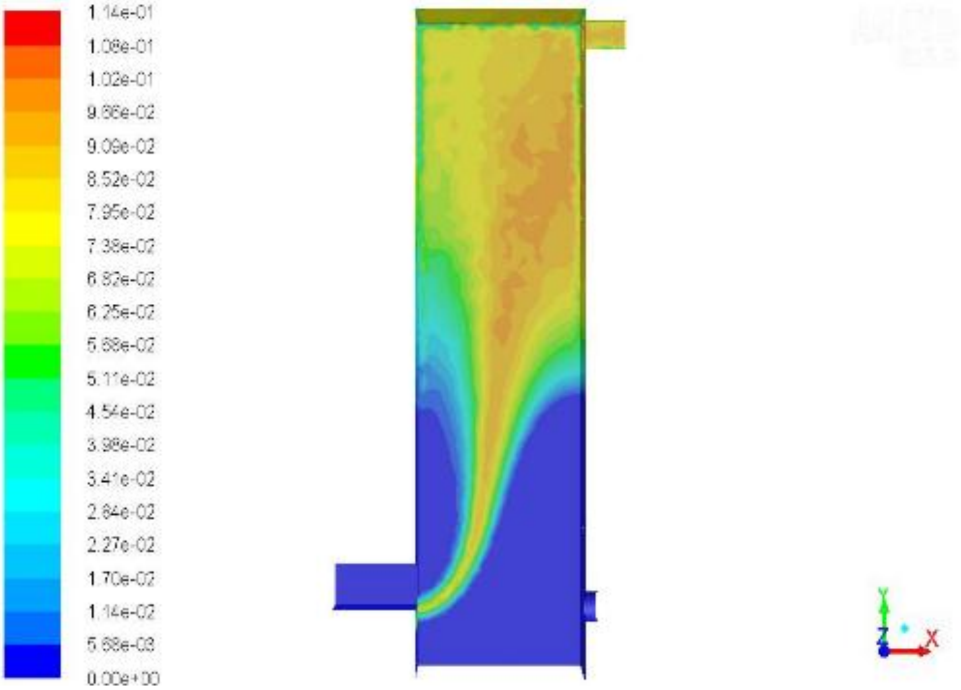

Figure3. *Contour plot of CO fraction*

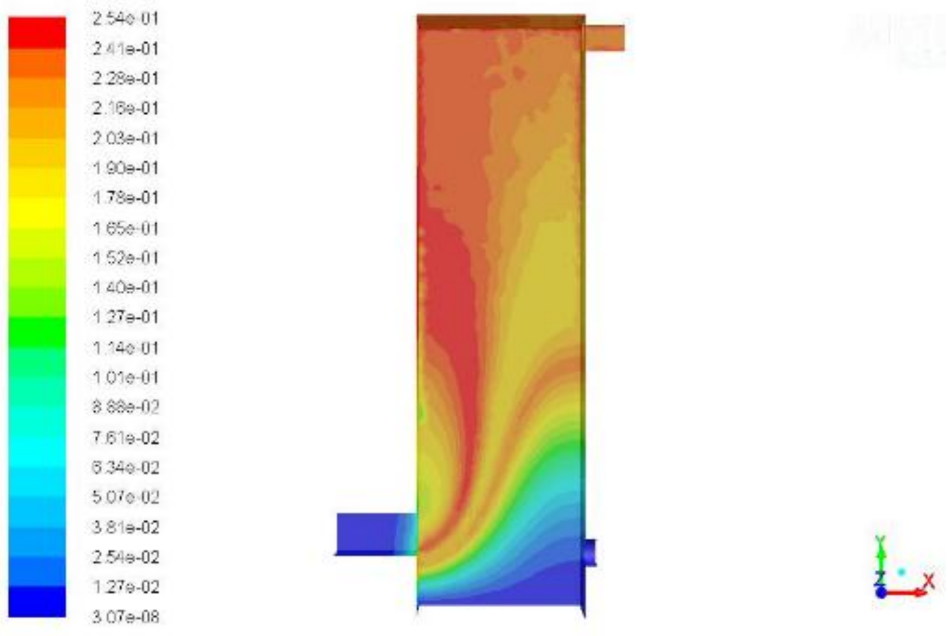

Figure4. *Contour plot of CO2 fraction*

Pollutants (SO_2 and NO_2) Mass Fraction

As in Figure 5, there is zero fraction of SO_2 in all part of the furnace. Maximums value of the SO_2 was found 0.00053 which is also negligible. So this result also satisfied the main application of the CFB which is low emission of SO_2.

NO_X emission is also very less. This is also required result for the CFB boiler. We know NO_X emission is not occurred in the CFB boiler because of lower combustion temperature. NO_X formation occurred at higher temperature that is more than 1200°C. Here maximum fraction of NO_X is 0.949 which is at the middle of the furnace this is because there is maximum temperature as shown in Figure 6.

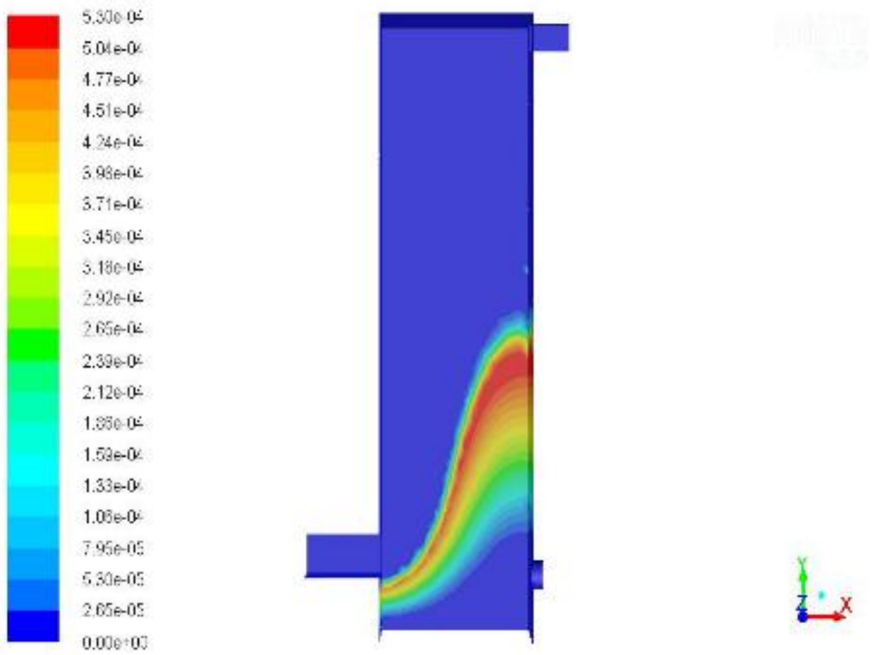

Figure5. *Contour plot of SO_2 fraction*

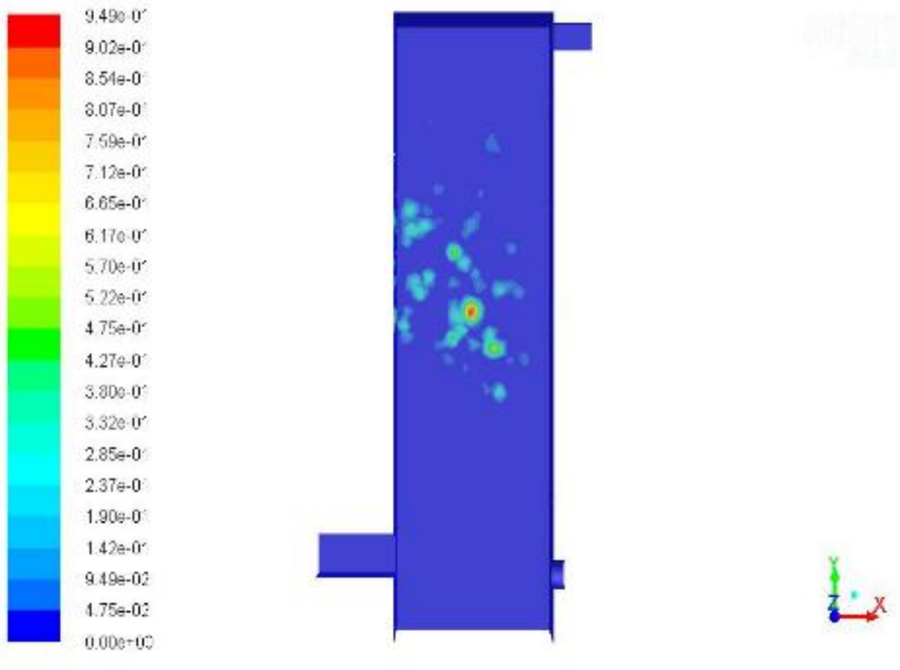

Figure6. *Contour plot of NO_X fraction*

Mixture Fraction and Density

Maximum mixture fraction is one which is at the inlet of rice husk and minimum mixture fraction is zero which is at the inlet of air. Inside the furnace the mixture of the fuel and air occurred so mixture faction is varied from 0.1 to 0.9. Mixture fraction according to Figure 7 is low at the lower part of the furnace because air is dominant there. But high in upper part of the furnace because air is combusted mostly but some fuel remains.

According to Figure 8 maximum density is at the inlet of the fuel because of small cross sectional area and minimum density is inside the furnace because air and fuel are disturbed all over the furnace.

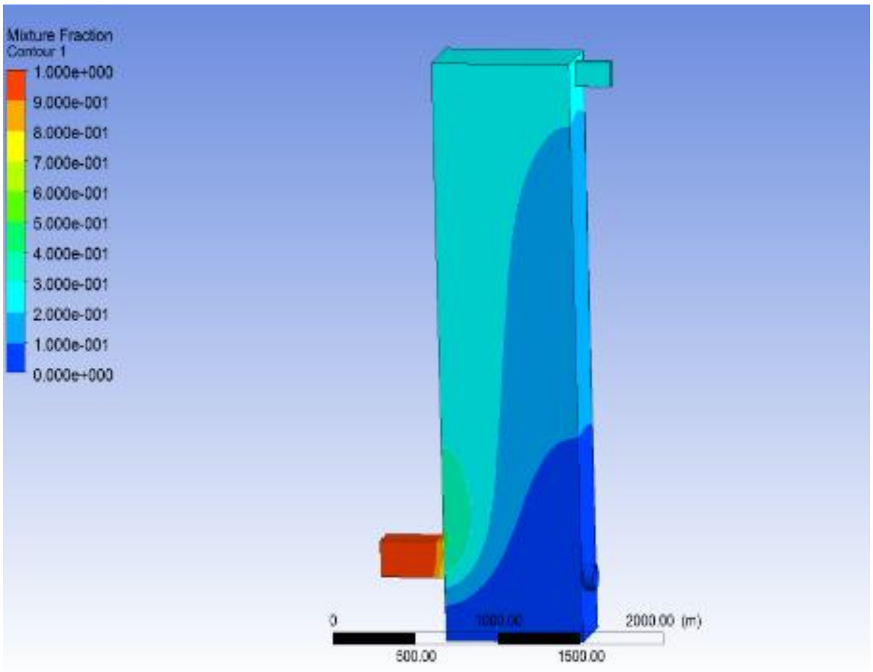

Figure7. *Contour of mixture fraction*

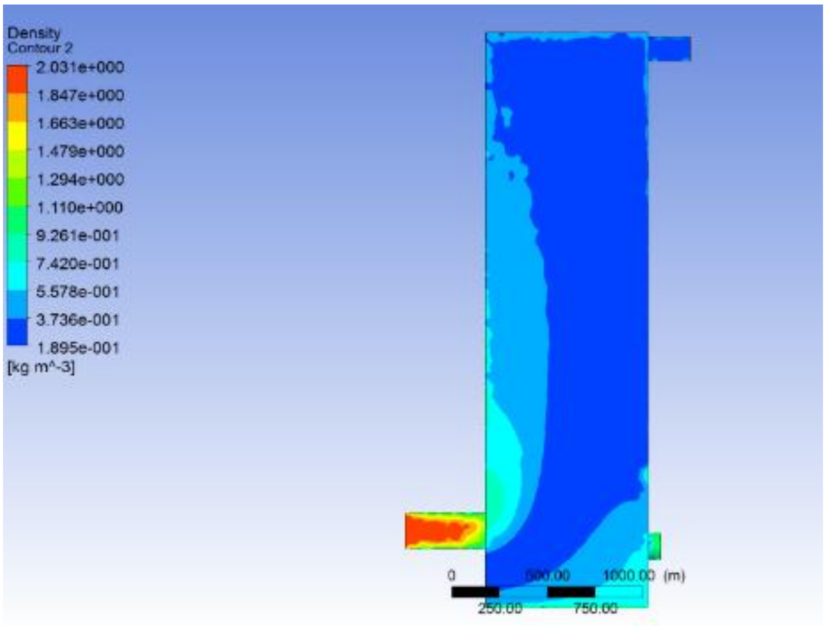

Figure8. *Contour of density*

CONCLUSION

On the basis of indirect method for efficiency calculation the design of a laboratory sized combustion chamber and a suitable hopper and cyclone separator for the required furnace was completed. To verify the circulating fluidized phenomenon the steady state simulation in ANSYS FLUENT 15.0 was conducted.

Hence, the objective of designing a fluidized bed combustion chamber along with is respective fuel hopper and cyclone separator has been fully accomplished.

ACKNOWLEDGEMENT

We would like to express our sincere thanks to Prof. Dr. Prabir Basu who helped us in the initial phases with providing us with reference material in fluidized combustion and our supervisor Dr. Bibek Baral, who let us experience the research in the vast field of combustion and patiently corrected our writing and supported our research.

We would like to express our deepest gratitude to our coordinator, Mr. Pratisthit Lal Shrestha, for his excellent guidance, caring, patience, and providing us with an excellent atmosphere for doing research. We would also like to thank our co-supervisors Mr. Pratisthit Lal Shrestha and Mr. Anup KC for guiding our research for the past year and helping us in design phase and to develop our background in finite element analysis and modelling along with constant guidance throughout the project

REFERENCES

[1] P. Basu, Combustion and Gasification of fluidized beds, Tyalor and francis, 2006.

[2] "Boilers & Thermic Fluid Heaters," *Energy Efficiency Guide for Industry in Asia*, 2006.

[3] J. HRISTOV, *FLUIDIZED BED COMBUSTION AS A RISK-RELATED TECHNOLOGY: A SCOPE*, pp. 1-12, 2001.

[4] KC, Lectures of Heat and Power Engineering, Dhulikhel: KU, 2014.

[5] Ansys, Ansys Fluent Tutorial Guide, Canonsburg: Ansys, Inc., 2010.

[6] H. Neopane, "Lecture Slides on Hydraulic Turbines," vol. I, 2013.