

Combustor sizing of an industrial boiler

Shreya Thakare¹, Rucha Pendse², Dr. A. P. Pandhare³

^{1,2} Student, Dept. of Mechanical Engineering, Smt. Kashibai Navale College Of Engineering, SPPU, Pune, India

³ Professor, Dept. of Mechanical Engineering, Smt. Kashibai Navale College Of Engineering, SPPU, Pune, India

Abstract—In this paper, we have analysed a CO fired boiler combustion process. We designed a combustor to meet the requirements of this power plant. Numerical calculation was validated using simulation software such as ANSYS CFX. We have come up with a possible way to reduce the cost of the combustor. The actual scenario is different from the calculated. Flow path, flow type, turbulence, proper mixing, all these things are taken into account for a software analysis. We have provided a potential area for saving material considering actual scenario.

Keywords— residence time, average velocity, Creo 3.0, Ansys CFX.

1. INTRODUCTION

By numerical analysis, we designed an combustor for given flue gas volume and temperatures. The fuel used is CO gas. The combustion air is used with 20% excess air for proper combustion. Modelling of the combustor (incinerator) was done in Creo 3.0. We validated the design in ANSYS –CFX for proper mixing and residence time(travel time) output.

By analysing the CFD results we aim to relate the results which we are getting are right and designed combustor is fit for the required conditions. Similarly we can also check whether the proper mixing of air and fuel is ensured with the designed dimensions.

1.1 Theory

The design of the combustor is done to check 2 parameters

1. Proper mixing and turbulence

Proper mixing of gaseous fuel and combustion air is validated. Entries of air and fuel are determined in order to get optimized turbulence in combustor.

2. Residence(travel) time

Time required by a particle to travel from the point of entry to the point of exit of combustor is known as residence or travel time. The residence time is checked using ANSYS-CFX

The optimal residence time, or working volume, needed to meet the required efficiency in the furnace is a function of many factors such as waste composition, waste characteristics, degree of turbulence, furnace operating temperature.

1.2 Numerical Calculation

Design is done assuming the residence time to be 0.9 seconds

.Q1 (capacity of flue gases) with 20% excess air = 59276.076 Nm³/hr

$$Q_1/T_1 = Q_2/T_2$$

.....{Temperature correction from NTP}

$$59276/273 = (Q_2)/((273+1305))$$

$$Q_2 = 342628.747 \text{ m}^3/\text{hr}$$

$$Q_2 = 95.174 \text{ m}^3/\text{sec}$$

$$Q_2 = A \times V$$

$$= \pi d^2/4 \times \frac{L}{0.9}$$

Assuming (L=3d)

$$= \pi L^2/36 \times \frac{L}{0.9}$$

$$95.174 = \pi L^2/36 \times \frac{L}{0.9}$$

Hence, L = 9.913 m

Length of the incinerator = 9.913 m

Diameter of the incinerator = 3.313 m

Velocity of gases in the incinerator = 11.033 m/s

1.2.1 Duct sizing-

Fuel discharge is 38054 Nm³/hr

At inlet of combustor, temperature of fuel is 200°C.

Hence by direct relation discharge at 200 °C is =

$$\frac{38054 \times 473}{273} = 65932.388 \text{ m}^3/\text{hr}$$

Fuel inlet duct carries a gaseous discharge of 65932.388 m³/hr. The velocity calculated experimentally is 15 m/s at the inlet. Hence, area can be calculated as below.

$$Q = A \times v$$

$$A = \frac{65932.388}{3600 \times 15}$$

$$A = 1.22097 \text{ m}^2$$

$$A = (\pi \times d^2) / 4$$

Hence,

$$d = 1.2468 \text{ m}$$

At the time of design we consider inner diameter = 1.2486m

And the thickness assumed = 16 mm

Hence, outer diameter = 1.2788m

Air Duct-

Discharge of air = discharge of fuel × air by volume

$$= 38054 \times 0.14832$$

$$= 5644.169 \text{ Nm}^3/\text{hr}$$

Inlet air is at 173° C

Hence discharge at inlet= 5644.169 × ((173+ 273))/273

$$= 9220.8773 \text{ m}^3/\text{hr}$$

$$Q = A \times v$$

$$A = 9220.8773 / (15 \times 3600)$$

$$= 0.170757$$

$$A = (\pi \times d^2) / 4$$

Hence,

$$d = 0.4663 \text{ m}$$

Inner diameter is 0.4663 m

And outer diameter is 0.498277 m

By this calculated information, we have created a model in Creo 3.0

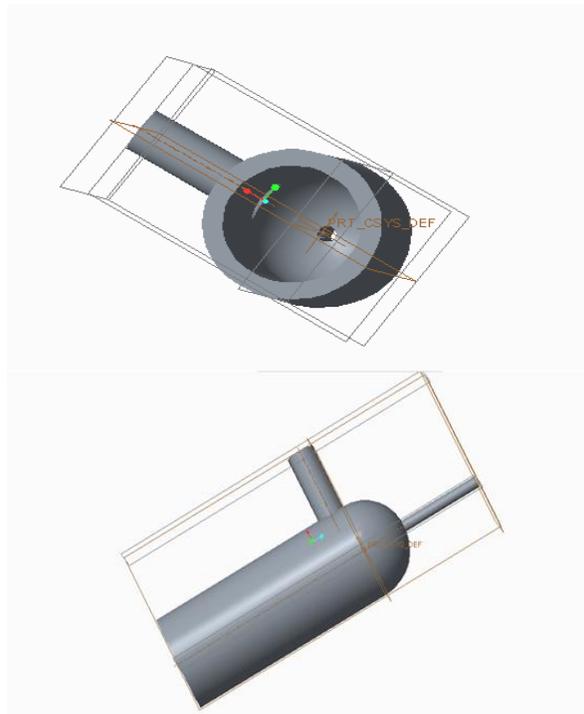


Figure 1.

2. SIMULATION IN ANSYS CFX

The model created in CREO 3.0 is imported in ANSYS workbench. For the designed combustor, radial entries could not incorporate proper mixing of gaseous fuel and air at given velocities, hence the fuel entry is kept tangential and air entry is kept axial to create the necessary turbulence.

For simulation in ANSYS, the imported model needs to be perfectly solid for better discretization. The imported model is given names at the inlets and outlets. The fluid specifications, their velocities, temperatures and different flow properties are assigned to each inlet and outlet duct.

2.1 Meshing

The partial differential equations that govern fluid flow and heat transfer are not usually amenable to analytical solutions, except for very simple cases. Therefore, in order to analyse fluid flows, flow domains are split into smaller subdomains (made up of geometric primitives like hexahedra and tetrahedral in 3D and quadrilaterals and triangles in 2D). The governing equations are then discretized and solved inside each of these subdomains.

Hence, the model is meshed accordingly into a number of elements. At curvatures, any irregularities or at any points where the load might be more, refinement of elements is done. These areas are further divided into finer elements in order to get more precise results.

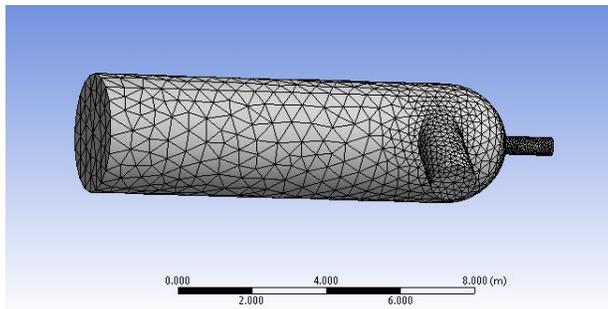


Figure 2

After meshing the fluid properties are defined After the solution is done, results can be determined using different parameters such as velocity, pressure, displacement, temperature etc.

Discretization done in our meshing is in tetrahedral form. The number of elements are 18874 and number of nodes 3978.

2.2 Velocity profile

In velocity profile, influence of velocity and position and trajectory of the flow of a liquid can be studied on another one. Overall velocity contours, profiles can be obtained. Magnitude of velocity at a specific point can be found out. In our analysis, we need to find the travel time of the particle. Hence we carry out the velocity profile as given below

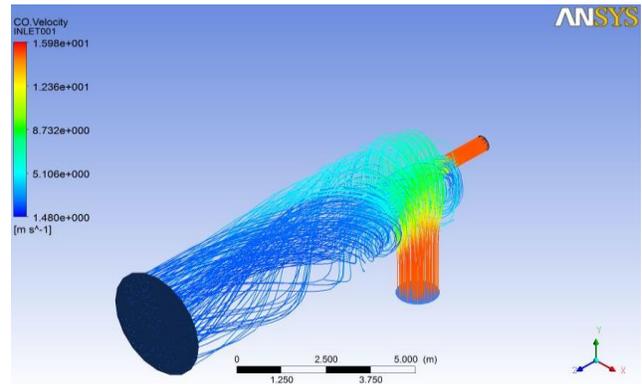


Figure 3

From the velocity profile,

It can be seen that the maximum velocity is 15.98 m/s

For a turbulent flow in the pipe,

$$V_{max} = V(1 + 1.33\sqrt{f})$$

Where f = friction factor

And V = average velocity

$$f = (100Re)^{-1/4}$$

Where, Re is the Reynold's no.

$$Re = \rho V D / \mu$$

Hence, Re calculated is 304721.39

$$f = 0.013459$$

$$V = 15.98 / 1.154299$$

Hence, $V_{avg} = V = 13.8438$ m/s

So, now for this velocity, less travel time is required

$$t = 9.93 / 13.8438$$

$$= 0.7172 \text{ sec}$$

Hence for this travel time, numerically length obtained would be

$$11.033 \times 0.7172 = \text{length}$$

$$\text{Hence length} = 7.91378 \text{ m}$$

Thus material required is reduced and hereby, the cost is decreased

3. CONCLUSION

From the calculations we can see that there is a slight variation in the travel time value of a particle. The calculated value is more than that we actually get in simulation. This is mainly due to the fact that a particle never travels linearly in a turbulent atmosphere. The trajectory in our case is spiral which increases the average velocity than the numerically designed value. So the variation in combustor design can be done to compensate for this reduction in time, which would result in reduction of length and material requirements, eventually decreasing the cost of the combustor.

units

Nm^3/hr = capacity at NTP

m/s = velocity

m^2 = Area

Equations

$$Q = A \times V$$

$$V_{\max} = V(1 + 1.33\sqrt{f})$$

$$f = (100\text{Re})^{-1/4}$$

$$\text{Re} = \rho V D / \mu$$

$$A = (\pi \times d^2) / 4$$

Table of figures

Sr. No	Figure Name	Page No.
1	Combustor modelling in Creo 3.0	02
2	Meshing in ANSYS CFX	03
3	Velocity profile	03

4. FUTURE SCOPE

In order to design the combustor for this boiler plant, we have assumed the residence time to be equal to 0.9 seconds. However, when we simulated the model in ANSYS CFX, the travel time obtained was less than 0.9. This was due to the turbulence created which increased average velocity reducing distance. New and changed design had to be made for the changed travel time. Hence, there was a scope to reduce the length of the combustor and hereby decreasing the material cost.

5. ACKNOWLEDGEMENT

We would like to express our gratitude to all the people who helped us put this paper into Publication. We also would like to thank Dr. A.P. Pandhare for constantly encouraging us during our project work.

REFERENCES

- 1) Cfd Analysis Of Waste Heat Boiler
Sreekanth S, Dr. Benny Paul
- 2) Cfd Modelling Of Waste Heat Recovery Boiler
M. Manickam, M.P. Schwarz, J. Perry