Analysis of Air Flow inside a Forced Draft Semi Gasifier Stove Using CFD Technique

Taha Kanchwala[1], N.R Gilke[2] and Indresh Tiwari[3]

[1]M.Tech CAD CAM Engineering, KJSCE
[2]Mechanical Engineering, KJSCE
[3]TATA POWER
E-mail: [1]taha.kanchwala@somaiya.edu, [2]nandkumargilke@somaiya.edu, [3]indresh.tiwari@tatapower.com

Abstract—*Poor thermal efficiency is obtained when solid fuels like biomass are directly combusted. Higher thermal efficiencies can be obtained if the solid fuel is converted into liquid or gaseous fuels. Gasification is one of the methods which can convert solid fuels into gaseous form directly in the presence of limited air, for indoor and domestic purposes semi gasification stoves are used. Gasification process can be optimized if proper amount of air is supplied to the biomass, the required airflow pattern can be studied by the help of Computational Fluid Dynamics(CFD). This paper presents the CFD analysis of the air flow pattern inside a forced draft semi gasifier biomass stove. The results obtained can be used for further improvement in the designing the stove and enhancing its efficiency.*

1. INTRODUCTION

India being a developing nation, were about 70% of the population lives in the rural part[1]. Thus it is necessary to work for the betterment of the rural areas. There are many aspects of life which needs a development. To begin with it was optimum to change the cooking practices. The rural India normally cooks their food by directly combusting the biomass, either in open areas or in their houses. Both practices create health hazards and have a poor efficiency of about 7%. According to WHO, "4.3 million people a year die prematurely from illness attributable to the household air pollution caused by the inefficient use of solid fuels"[2].

The Indian government subsidizes LPG and kerosene to keep their prices low, this policies discourages the practice of dismantling of forests for cooking fuel. Due to economical and political issues, people are unable to take benefits from Government schemes and policies, and they continue using biomass as their primary fuel.

Biomass burning stoves are available in the market, but the issue of health hazard and poor efficiency remains. On investigation of such stoves it observed that major issue with the biomass stoves was incomplete combustion, which is due to lack of oxygen for combustion. Semi Gasifier stoves, which are a type of biomass stove are facing the same issue. To find a solution for better flow pattern of air in the existing semi gasifier stove, CFD technique could be used. CFD will help to understand the flow of air inside the stove.

Semi-gasifier stoves are a type of portable biomass stove which are usable for indoor purposes. They are designed either using natural draft or forced draft. Natural draft has a lower efficiency and forced draft designs need a fan assembly to be installed. The main designing part of a semi gasifier stove includes the design of it height, diameter and holes for air passes, the design for gasifier stove can be found in Belonio's handbook[3,4]. As per the rules of gasification we provide about 30% of air required for total combustion for solid fuels to gasify[5]. This arrangement is made by providing holes in the primary and secondary chamber of the stove body as shown in Figure 1 below. The bottom or primary holes are responsible for providing air for gasification and the top holes or secondary holes are responsible for providing air for combustion.

The CFD analysis will help us in understanding the velocity and the pressure flow[6] inside the semi gasifier stove. This will help to decide the arrangements of the holes in the semi gasifier stove to achieve proper gasification and simultaneously combust the gases. Practically it is difficult to measure the pressure inside the stove. This difficulty is overcome by using virtual modeling and analyses. The FLUENT module in ANSYS is used for analyzing the air flow pattern of the stove.

2. DESIGN OF THE STOVE

There are many parameters of gasifier stove, we need to give proper considerations during the design process to achieve the maximum performance of the stove. The following factors were considered for gasifier stove design.

height of cylinder =300mm

Diameter of the outer cylinder= 150mm

Diameter of the internal cylinder=140mm

The inner cylinder in the stove is also termed as reactor. This is the component were the fuel is placed and made to be burned in limited amount of air. The passage of the air in the cylinder is normally done by 2 sets of holes, i.e. the Primary holes(bottom holes) and the Secondary holes(top holes).

The Primary holes are responsible to provide limited amount of air for gasification of biomass when the stove is in working position, whereas the secondary holes provide the necessary air for the burning of the gases in the top portion, were the utensils can be kept.

CFD technique will help us in understanding the position of the holes and also will give us an idea how different diameters of hole will change the flow pattern of air inside the stove cylinder.

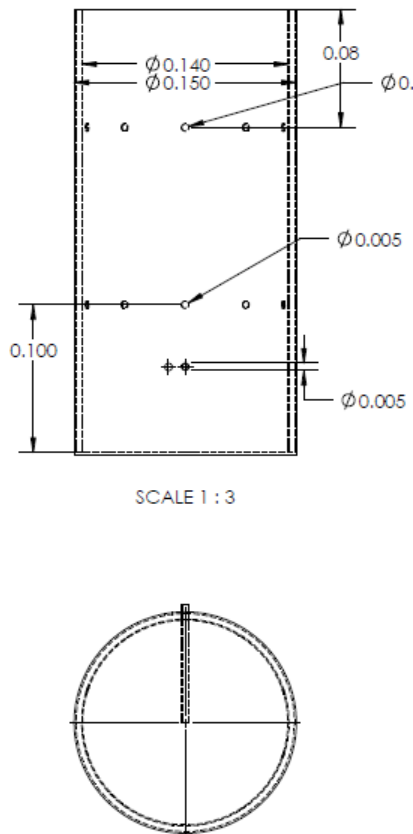The design of the stove is shown in Figure 1.

Figure 1. Stoves Design

3. CFD ANALYSIS ON STOVE

The CAD model was made in Solid Works software and saved it in .igs format , further CFD analysis was carried out on ANSYS fluent module with k-ε model . The analysis works on 3 major steps, Pre-processor, Solver, Post-processor[7].

3.1 Pre-processor

On observation it was noticed that instead of cad model of the stove we should model the air flow part of the stove which will help us by using less computational time, and quicker results. Thus the air flow part was modeled. The materials, boundary condition and the equations to be solved were given as the input. The components were defined, such as, inlet and outlet. After which pre-processor created the mesh for given design, as the mesh is an important attribute in CFD solver, finer mesh was selected for accurate results.

3.2 Solver

Once the pre-processor has all the values in it, the solver starts solving the equations on the mesh generated. The results were converged after 700 iterations.

3.3 Post-processor

The solved results were viewed in the post-processor section different profiles, contours, vectors, animations were analyzed.

4. RESULTS

Fifteen   iterations were made by changing the boundary conditions. From the results obtained we were able to decide and change the design parameters of secondary and primary holes. This decision was based on the flow pattern obtained from FLUENT, the pressure profile and the velocity profile. The complied data can be seen in Table 1 below.

Table No.1 Compiled Results

Sr. no	Air input (Size (mm))	Primary holes (no.,Diameter (mm))	Secondary holes (no.,Diameter (mm))	Pressure range (pascal)	Velocity range (m/s2)
1	Ø 5	10, Ø 5	10,Ø 5	-8.4e+8to 4.5e+4	0 to 3.8e+4
2	Ø 20	10,Ø8	10,Ø8	-7 to 151	0 to 15.7
3	Ø 20	10,Ø10	10,Ø10	-7.3 to 177	0 to 17.5
4	Ø 20	10,Ø12	10,Ø12	-6.7 to 170	0 to 17
5	□ 20	10,Ø8	10,Ø8	-13.1 to 160	0 to 19.6
6	□ 20	10,Ø10	10,Ø10	-15.6 to 160	0 to 20.3
7	□ 20	10,Ø12	10,Ø12	-13.3 to 154	0 to 19.1
8	□ 20	5,Ø8	10,Ø8	-14.6 to 201	0 to 22.8
9	□ 20	5,Ø10	10,Ø10	-16.9 to 172	0 to 21
10	□ 20	5,Ø8	10,Ø8	-107 to298	0 to 19.5
11	□ 20	5,Ø10	10,Ø10	-108 to 216	0 to 20.3
12	□ 20	5,Ø10	10,Ø8	-14.4 to 183	0 to 19.7
13	□ 20	5,Ø10	10,Ø8	-10.5 to187	0 to 20.3
14	□ 25	5,Ø10	10,Ø8	-67.4 to 118	0 to 26
15	□ 25	5,Ø10	10,Ø8	-91.4 to 216	0 to 19.3

The results of the first iteration shows that the maximum pressure and velocity in the stove will be 8.4e+8pascal and

80000m/s. refer Figure 2 and Figure 3, these results are high for a stove to withstand. Therefore the size of the inlet of the stove, primary and secondary holes was changed. With these changes the second iteration was made with a air inlet vent of Ø 20mm, the result obtained were acceptable as the maximum pressure and maximum velocity was 151 pascal and 15.7 m/s. refer Figure 4 and Figure 5. But the flow pattern was propagating in forms of concentric circles, because of which the pressure distribution amongst the holes was not uniform. Furthermore two more iterations were made to solve the issue of uniform flow, but the pattern of concentric circles remains same in circular inlet of the stove.

Therefore in fifth iteration square cross pipe of side 20mm was selected for next iterations, the flow was in a whirl pattern and the pressure at primary and secondary holes was seen to be equal. The results for the fifth iteration are shown in Figure 6 and Figure 7. To check effect of different hole size and inlet size more iterations were performed. from which the result of iteration 15 was observed to be the most suitable for designing the stove as it gave the required pressure and velocity of air for better circulation. The pressure profile and velocity profile are shown in Figure no. 8 and Figure no. 9 respectively.
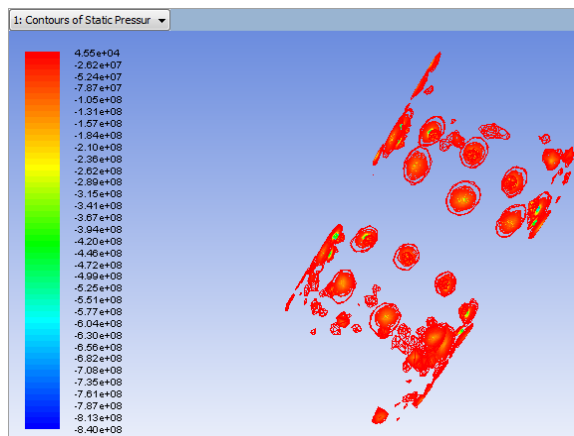

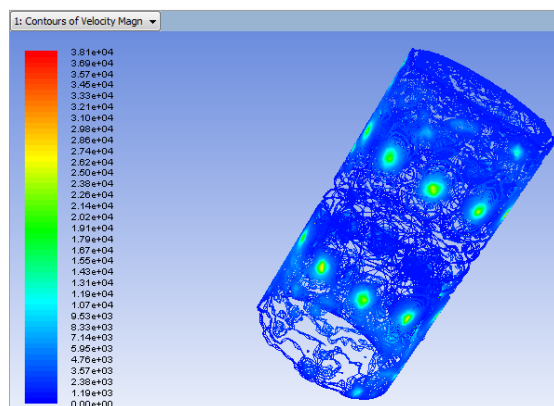**Figure 2. Iteration 1 pressure profile**
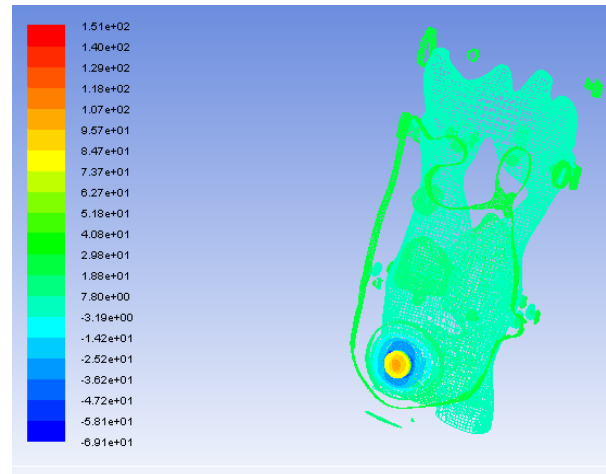

Figure 3. Iteration 1 velocity profile

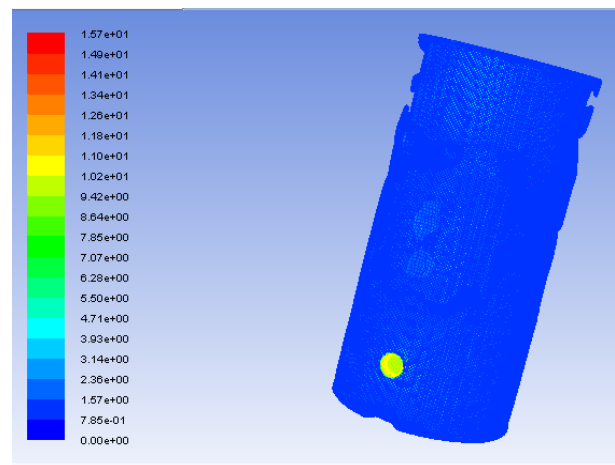**Figure 4. Iteration 2 pressure profile**

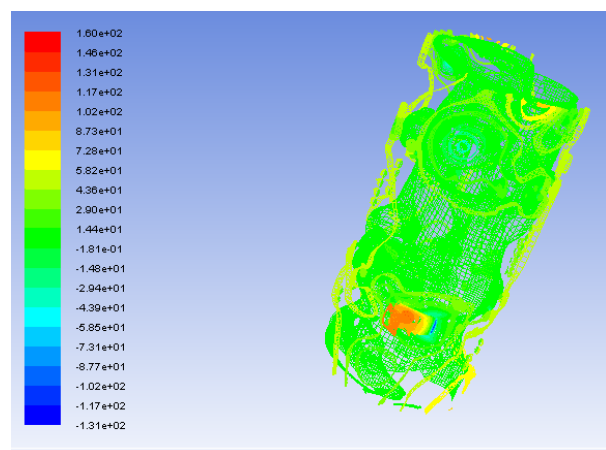**Figure 5. Iteration 2 velocity profile**

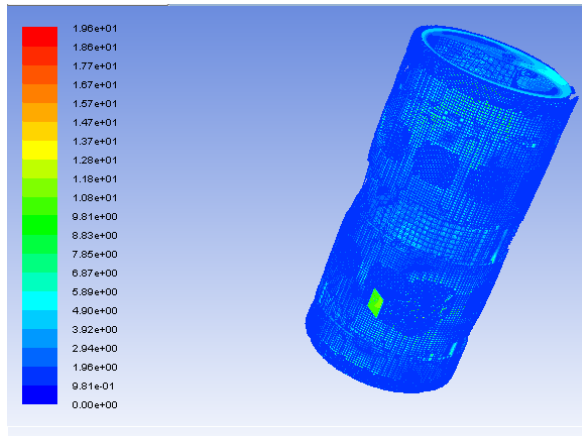**Figure 6. Iteration 5 pressure profile**

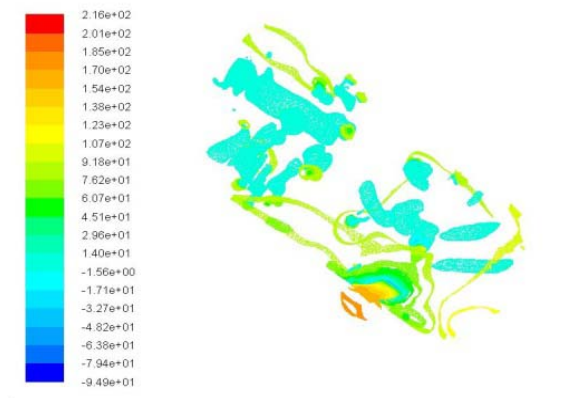**Figure 7. Iteration 5 velocity profile**

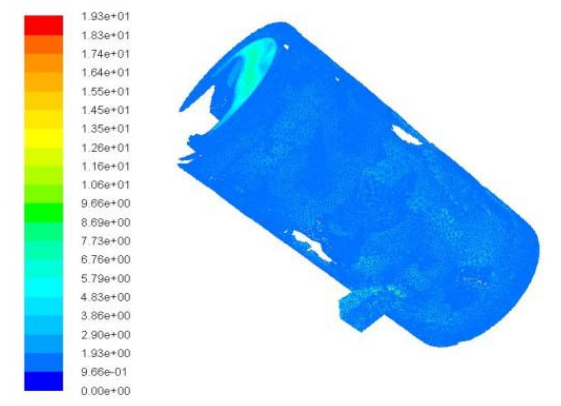**Figure 8. Iteration 15 pressure profile**

Figure 9. Iteration 15 velocity profile

4.1 Discussions

Each set of results were analyzed on the bases of the flow path, pressure profile and velocity profile. The analyses were mainly done on the activeness of the holes, i.e. is air flowing through the hole or not?

While analyzing it was noticed that if all the holes either primary or secondary are kept of Ø12mm then one or more holes were missed . So on further analysis the Ø12mm holes were eliminated. Similarly it was notices that the holes get back pressure at the inlet if they were of larger diameters, thus it was decided to check the holes of < Ø10mm. The back pressure effect was less in these holes. Similarly when small holes of size Ø5mm were analyzed, the pressure recorded was too high, and abnormal flow pattern was seen.

The air inlet section of the stove when kept circular showed that the air flows in a form of concentric circles, which made the pressure distribution amongst the holes uneven. thus on a trial, a square cross section of air inlet was decided and analyzed, a whirl flow pattern was seen and it also kept the pressure amongst all holes same.

4.2 Validation

Virtual Testing was done by CFD Analysis, this testing was carried out to study the air flow pattern through the primary and the secondary holes of the reactor. The results were validated by practical experimentation.

5. CONCLUSION

The forced draft semi gasifier stove has an efficiency of about 35% with CO <5 g/MJd as described by MNRE. This helps by giving a cleaner fuel and reducing the devastating effects of CO to the environment and the human health. The Gasification of biomass can be maximized by providing the correct amount of air to gasify, this was achieved by providing a good structural design of the stove with proper implementation and use of primary and secondary holes. This improved performance of the stove is achieved by implementing the use of virtual prototyping and CFD. CFD made it possible to analyze the annulus section of the stove, which otherwise would be difficult to be done.

REFERENCES

[1] Komala H. P. And Devi Prasad A. G., Biomass: A Key Source Of Energy In Rural Households Of Chamarajanagar District, *Advances in Applied Science Research*, 7(1):85-89, 2016

[2] World Health Organization(WHO) Statistic,// http://www.who.int/mediacenter/factsheets/fs292/en/

[3] Alexis T. Belonio, Rice Husk Gas Stove Handbook,2005

[4] Khadija Sultan and Munir Ahmad, Energy Efficient Biomass Gasifier Stove for Domestic Purpose, *Journal of energy technologies and policy*, Vol.5, No.10, 2015

[5] Prabir Basu, Biomass Gasification And Pyrolysis, *Elseiver Inc*, August 2010.

[6] Md. Tanvir Sowgath, Md. Mominur Rahman, Sabbir Ahmed Nomany, Md. Nazmus Sakib, Md. Junayed, CFD Study of Biomass Cooking Stove using Autodesk Simulation CFD to Improve Energy Efficiency and Emission Characteristics, *The Italian Association of Chemical Engineering*, 10.3303/CET1545210

[7] John D. Anderson Jr., Computational Fluid Dynamics, McGraw-Hill International Editions, 1995.