Alganash, B.S., Paul, M.C. and Watson, I.A. (2011) *Experimental and computational investigations of biomass mixing and combustion processes.* In: 5th European Combustion Meeting, 28 June - 1 July 2011, Cardiff University.

Experimental and Computational Investigations of Biomass Mixing and Combustion Processes

Blaid S. Alganash, Manosh C. Paul*, Ian A. Watson
Systems, Power & Energy Research Division, School of Engineering, University of Glasgow, Glasgow, G12 8QQ, UK

Abstract

Biomass is considered to be a promising source of sustainable energy and consequently it is attracting more research attention. The focus of this ongoing study is on the performance of biomass combustion processes. Computational fluid dynamics (CFD) techniques are planned to simulate the combustion process in various models. This is done in parallel with experimental work to understand the mixing and combustion behaviour biomass. For this purpose CFD models based on both Euler-Lagrange and Euler-Euler have been used. A new combustion chamber has been designed and fabricated to validate the theoretical models. As a first step research investigations on static combustion are currently underway. The mixing and concentration of a small fixed quantity of carbon powder has been investigated and some preliminary results are presented.

1. Introduction

Biomass is considered to be a promising source of sustainable energy and has a significant role in producing energy for industrial and domestic use. There have been many researches into biomass as a renewable energy source. The driving force and motivation for this is the environmental awareness related to the emission of greenhouse gases and the concerns about the decreasing resources of fossil fuels. Biomass fuels can be utilised in different ways such as combustion, gasification, and co-combustion.

The combustion of solid particles of biomass is a complex process. It involves fluid dynamics, heat transfer and chemical kinetics. Moreover, it takes place through a number of stages consisting of consecutive homogeneous and heterogeneous reactions. Investigations of the behaviour of biomass combustion are fundamental to all practical applications. Di Blasi [1] developed a detailed particle energy and mass transport model to investigate the influence of physical properties of biomass devolatilization. It was concluded that biomass density and char thermal conductivity exhibit the highest sensitivity. Yang et al [2] studied the combustion characteristics of a single biomass particle. The range of particle size was (10µm-20mm). A CFD model using the discrete phase model (DPM) was developed and successfully applied for the numerical predictions of pulverised wood combustion. Certainly, the size of particles plays a fundamental role in combustion and gasification processes. It is easy to imagine that smaller particles carried by the gas stream tend to be consumed faster and more easily than the larger ones. Therefore, the particle size distribution influences not only the rate at which the fuel reacts with oxygen and other gases, but also all most all other aspects of combustor and gasifiers. The effects of particle size, reactor heating, and reactor temperature were theoretically and experimentally investigated by Di Blasi [3]. It was found that char yields increase as particle size increases. In addition, higher heating rates leads to higher volatile matter yields and lower char yields. Park et al. [4] investigated heat and mass transfer processes of 25.4 mm diameter dry wood spheres experimentally and theoretically. Comparison between the theoretical model and experiments showed good agreement with both the temperature and the solid mass loss measurements. Miltner et al. [5] investigated the maximisation of the thermal output of new combustion chamber by using CFD. The generated results gave an outline of how CFD is useful in the field of biomass combustion and can be used to optimise the design of combustion chamber and the operational parameters.

Undoubtedly, the significance of comprehensive, multidimensional combustion models developed by using CFD is clear. An overview of combustion modelling was presented in [6-8]. In terms of dispersed two phase flow, there are two approaches applied in CFD which are Euler-Lagrange and Euler-Euler. In the former one the gas phase is described by Eulerian and the particulate phase is by a Lagrangian frame reference, but in the latter one the two phases are modelled adopting the Eulerian description. A three-dimensional large eddy simulation (LES) was applied to investigate the interactions between the gas and solid phases [9]. Good results were obtained concerning the evaporation and heat release conditions and the description of the solid fuel particles. Numerical investigation of wood gasification in a fluidised bed reactor by using Euler-Euler approach was carried out by Oevermann et al. [10]. It was shown that the results were partially in agreement with experimental data. Compared to Euler-Lagrange and direct numerical simulation models, the Euler-Euler approach is computationally less demanding. The same approach was employed in modelling the dispersed two-phase flow in pulverized coal combustion [11]. It was shown that this model

* Corresponding author: Manosh.Paul@glasgow.ac.uk
Proceedings of the European Combustion Meeting 2011

provides a comparable overall accuracy compared to the Eulerian-Lagrangian approach. In this ongoing research study CFD is applied to various models to simulate the combustion process, and a new combustion chamber was designed and fabricated to validate these models. The aims and the main objectives are to understand the combustion behaviour and how important parameters such as temperature, particle size, moisture content, pressure and so forth affect the combustion process. Moreover, the aim is to investigate the interactions between the phases (liquid, gas and solid phases). Ultimately, the goal is to achieve a suitable combination of parameters that would potentially reduce emissions and maximise the thermal output.

The current investigations are related to the mixing and combustion of a fixed quantity of carbon powder in particulate form in the chamber. The solid carbon–air mixture is dispersed in the chamber by a controlled blast of air. This will then be ignited by using an electrical spark and other forms of ignition such as laser. Note an experimental study was carried out by Dubaniewicz et al. [12] to ignite coal dust in air by using a continuous wave laser beam.

2. Experimental work and equipment

2.1 Design of the combustion chamber

The combustion chamber has been designed using Solid works. A small pipe with three holes each 3 mm diameter is inserted inside the chamber to intake the air flow into the system. A threaded rod is used in the centre position to hold a small cup for biomass and designed in a way that it could move up or down. The chamber is shown in Figure 1.

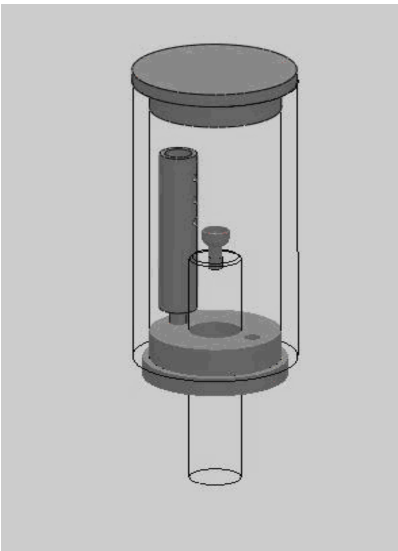


Fig. 1. Combustion chamber

A frame (Figure 2) to hold the top and the bottom caps was designed to secure the combustion chamber, made from fused quartz, during the experimental work to a table.

2.2 Fabrication and materials

All the parts of the chamber were manufactured in-house. The materials used are steel and stainless steel.

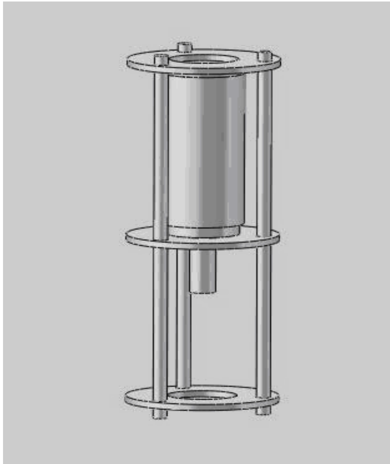


Fig. 2. Combustion chamber with holding frame

All the parts have been assembled and fixed in a place where the experimental work takes place. A cylinder of oxygen has been fitted to supply the oxygen and an electronic ignition source has been provided at the top of the combustion chamber to generate the electrical spark. Most recently, a couple of trials have been carried out to ignite the solid carbon particles but due to the time constraints it was not possible to calibrate the experimental data. Thus, no experimental results are presented in this paper. Only the CFD results of the biomass mixing with air are shown and the methodologies used in the numerical simulations are briefly explained in the section below.

3. CFD modelling

3.1 pre-processing

The geometry created by solid works has been exported to the geometrical pre-processor GAMBIT [13] to generate the mesh inside the combustion chamber. The mesh has been generated for four connected volumes (the main domain, the small cup where the carbon powder is placed, the rod that holds the cup, and the pipe with the three hols), Figure 3 illustrates the geometry of computational domain.

3.2 Two phase modelling

The turbulence has been modelled by using the κ-ε dispersed turbulence model. This model is suitable when concentration of the secondary phase is dilute [13]. The model constants used for this simulation are the default ones in Fluent [13].

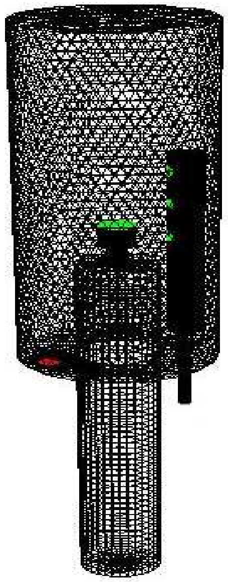


Fig. 3. Geometry of computational domain

3.2.1 Some assumptions and boundary conditions

The Eulerian multi phase model has been used to simulate the flow without combustion in the present case. It solves the momentum and continuity equations for each phase. To account for the maximum packing limit the granular option was activated. The solid carbon particles are assumed to be spherical in shape with a diameter of $1x10^{-5}$ m (the default value). The virtual mass effect is neglected because the density of the solid phase is greater than that of the gas phase. Since the particle size is small the lift force is not significant and as a result it has been neglected. Hence, the interaction between the phases is only due to the drag force. The maximum packing limit has been set to 0.63. The inlet air gauge pressure was 1 bar.

3.2.2 Governing equations

Gas phase:

The continuity equation for the gas phase (air) is written as

$$\frac{\partial}{\partial t}\left(\alpha_g \rho_g\right) + \nabla.\left(\alpha_g \rho_g \vec{v}_g\right) = 0 \qquad (1)$$

where $\vec{v}_g$ is the velocity of gas phase and α_g and ρ_g are the volume fraction and density of the gas phase respectively.

The momentum equation is written as

$$\frac{\partial}{\partial t}\left(\alpha_g \rho_g \vec{v}_g\right) + \nabla.\left(\alpha_g \rho_g \vec{v}_g \vec{v}_g\right) = -\alpha_g \nabla p + \nabla.\overline{\overline{\tau}}_g + \alpha_g \rho_g \vec{g} + K_{sg}\left(\vec{v}_s - \vec{v}_g\right) \qquad (2)$$

where $\overline{\overline{\tau}}_g$ is the gas phase stress-strain tensor, $\vec{g}$ is the acceleration due to gravity and K_{sg} is the momentum exchange between the gas and solid phases.

Solid phase:

The continuity equation for the solid phase is as follows

$$\frac{\partial}{\partial t}\left(\alpha_s \rho_s\right) + \nabla.\left(\alpha_s \rho_s \vec{v}_s\right) = 0 \qquad (3)$$

Where $\vec{v}_s$ is the velocity of solid phase and α_s and ρ_s are the volume fraction and density of the solid phase respectively.

The momentum equation is

$$\frac{\partial}{\partial t}\left(\alpha_s \rho_s v_s\right) + \nabla.\left(\alpha_s \rho_s \vec{v}_s\right) = -\alpha_s \nabla p_s - \nabla p_s + \nabla.\overline{\overline{\tau}}_s + \alpha_s \rho_s \vec{g} + K_{gs}\left(\vec{v}_g - \vec{v}_s\right) + \left(\vec{F}_{ex,s} + \vec{F}_{lift,s} + \vec{F}_{vm,s}\right) \qquad (4)$$

where $\overline{\overline{\tau}}_s$ is the solid phase stress-strain tensor, p_s is the solid pressure, $K_{gs} = K_{sg}$ is the momentum exchange coefficient between the solid and fluid phases; and $\vec{F}_{ex,s}$, $\vec{F}_{lift,s}$ and $\vec{F}_{vm,s}$ denote the external body force, lift force and virtual mass force of solid phase respectively.

The interspatial fluid-solid momentum exchange coefficient (K_{sg}) which includes drag coefficient and relative Reynolds number between the phases are calculated by using the Syamlal-O'Brien model [13].

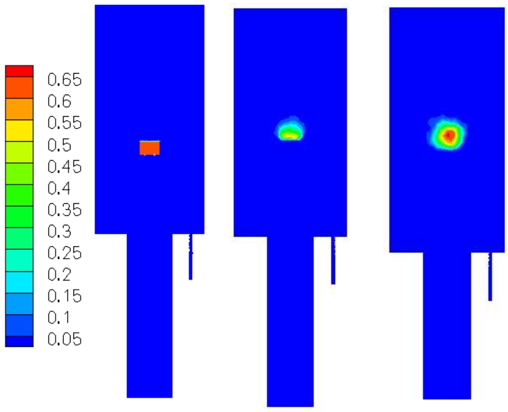
4. Simulation results

When solving the governing equations in fluent, the convergence criterion for the residuals of the all equations has been set to 10^{-3}. As iterations carried out, the residual monitors are tracked and the solution convergence has been checked.

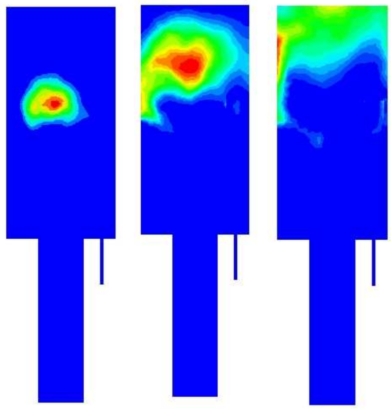
In Figure 4 the volume fraction of solid phase at different times is shown on the vertical middle plane (X-Y) of the combustion chamber. To ignite the mixture of the air and solid carbon particles it is important that a sufficient amount of dust concentrations is used. The results shows that the concentration level of solid carbon lifts from the cup as the time progresses and the presences of carbon dust close to the top part of the chamber would ensure the combustion sustainability, but this needs to be investigated further. Based on the combustion performance achieved the design of the chamber will be optimised. Amount of carbon and the controlled blast of air which passed through the three holes fitted in the pipe inside the chamber would also be varied and their effects will be investigated.

5. Conclusion

To sum up, the biomass contribution in the world energy supplies has become important due to concerns about greenhouse emissions and the depletion of fossil fuels. The initial results of the present work give an overview of how CFD can be used to simulate the mixing of biomass, in this case carbon powder. Investigations are still underway to the process of carbon combustion computationally. Then, these results will be used as inputs for the experimental work to validate the models.



(a) t = 0 sec (b) t = 0.001 sec (c) t = 0.003 sec



(d) t = 0.004 sec (e) t = 0.005 sec (f) t = 0.006 sec

Fig. 4. Volume fraction of solid phase with time

6. References

1. C. Di Blasi, Influences of physical properties on biomass devolatilization characteristics, Fuel, 76 (1997) 957-964.
2. Y. B. Yang, V. N. Sharifi, J. Swithenbank, L. Ma, L. I. Darvell, J. M. Jones, M. Pourkashanian, A. Williams, Combustion of a single particle of biomass, Energy & Fuels, 22 (2008) 306-316
3. C. Di Blasi, kinetics and heat transfer control in the slow and flash pyrolysis of solids, Ind. Eng. Chem. Res., 35 (1996) 37-46.
4. W. C. Park, A. Atreya, H. R. Baum, Experimental and theoretical investigation of heat and mass transfer process during wood pyrolysis, Combust and Flame 157 (2010) 481-494.
5. M. Miltner, A. Makaruk, M. Harasek, A. Friedl, CFD-modelling for the combustion of solid bald biomass, Fifth International Conference on CFD in the Process Industries CSIRO, Melbourne, Australia, (2006).
6. A. M. Eaton, L. D. Smoot, S. C. Hill, C. N. Eatough, Components, formulations, solutions, evaluation, and application of comprehensive combustion models, Progress in energy & combustion science, 25 (1999) 387-436.
7. C. Di Blasi, Modelling chemical and physical processes of wood and biomass pyrolysis, Progress in energy & combust, 34 (2008) 47-90.
8. Y. Wang, L.Yan, CFD studies on biomass thermo-chemical conversion, Int. J. Mol. Sci, 9 (2008) 1108-1130.
9. R. Kurose, H. Makino, Large eddy simulation of a solid-fuel jet flame, Combust and flame, 135 (2003) 1-16.
10. M. Oevermann, S. Gerber, F. Behrendt, Numerical simulation of wood gasification in a fluidized bed reactor using Euler-Euler modelling, Proceedings of the European combustion meeting, (2009).
11. A. C. Benim, B. Epple, B. Krohmer, Modelling of pulverised coal combustion by a Eulerian-Eulerian two phase flow formulation, Progress in Computational Fluid Dynamics, 6 (2005) 345-361.
12. T. H. Dubaniewicz, K. L. Cashdollar, G. M. Green, Continuous wave laser ignition thresholds of coal dust clouds, J Laser Appl. 15 (2003) 184-191.
13. Fluent 6.3 User's Guide. (2005).