World Applied Sciences Journal 19 (1): 100-105, 2012
ISSN 1818-4952;

DOI: 10.5829/idosi.wasj.2012.19.01.2756

Numeric Simulation of Turbulent Flows in Cylindrical Swirl Combustion Chamber Designed for Fuel Based on Coal Enrichment Wastes

I.A. Kuksov, S.P. Mochalov and V.D. Sarychev

Siberian State Industrial University (SibSIU), Novokuznetsk, Kirova St, 42, Russia

Abstract: The article examines the theoretical and practical aspects of modeling of gas-dynamic processes in the swirl chamber. The results of numerical computation of a swirling gas flow induced by two-row header in cylindrical chamber are presented. The description of two-phase flow model subject to the injection of particles is given. It is revealed that in stationary mode two characteristic zones are formed: axial zone of quasi-solid rotation and cylinder surface zone.

Key words: Swirling flow · Two-phase flow · Swirl chamber · Couette-Poiseuille flow · CFD simulation

INTRODUCTION

The analysis of the aerodynamic structure is of considerable interest to identify the advantages and disadvantages of combustion devices. Combustion reaction is preceded by a stage of mixing the oxidizer and fuel, so it's important to have an idea of the turbulence degree in the combustion space. The main effect of swirl is the improving of flame stability [1].

One of the commonly used designs of combustion chambers is a swirl combustion chamber. There are two main types of swirl combustion chambers. In the chamber of the first type a mixture of air and fuel swirl before it enters the working chamber, where combustion takes place. In the chamber of the second type air is supplied tangentially into the chamber and then is mixed with fuel supplied through separate channels.

Also, the location of the air and fuel inlets vary. They can be located in the side walls of the chamber or at the bottom opposite the outlet channel for the combustion products.

As it is shown in [2] the dimensionless values like size of the swirl core, the static pressure difference in the chamber and the radial distribution of static pressure functionally dependent only on the geometry of the chamber.

In such chambers mixture formation goes faster and the increase the chamber effectiveness is possible due to more complete fuel combustion. In addition, swirl chambers due to the centrifugal effect will increase the degree of slag catching as slag particles have a bigger density.

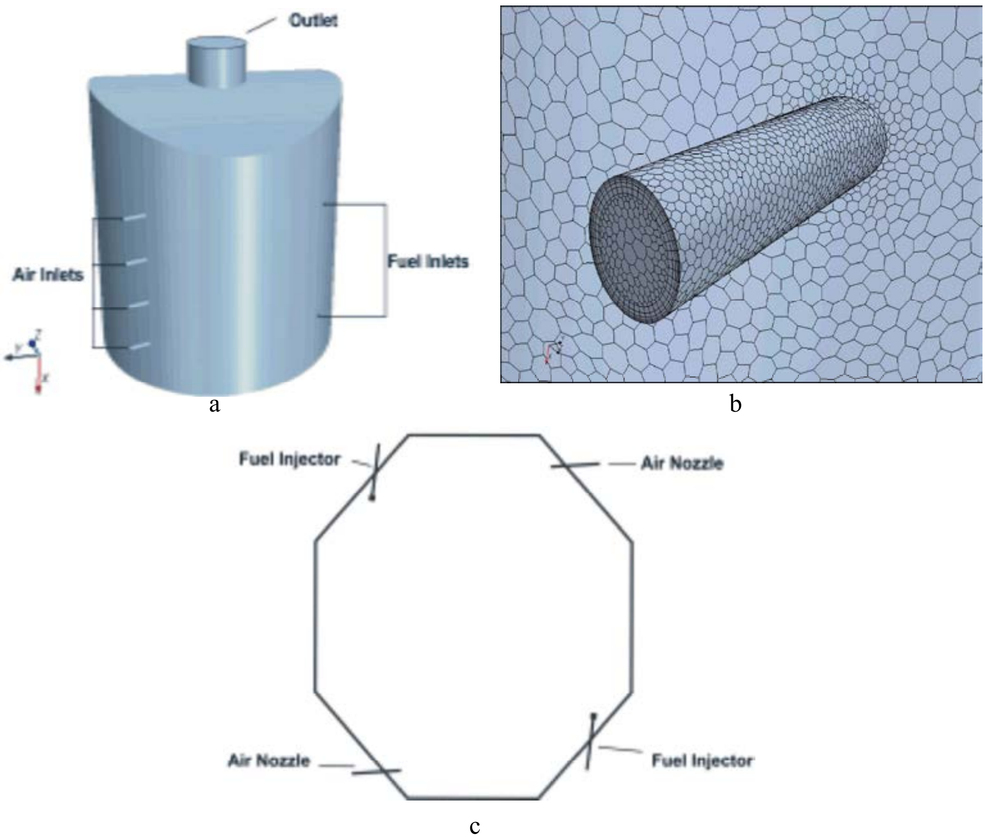
In this article the combustion chamber with tangential air supply through the nozzles in the side walls of the chamber is examined. At the same time the compressed air supplied along with fuel through a centrifugal injector is used to inject the fuel.

Flow swirling in a centrifugal chamber, carried out by means of a tangential gas inlet through the channels that are located in different ways around the perimeter, were investigated in detail in several papers. They found a strong connection between the rotating flow characteristics and the camera geometry, the location of feeders and a diameter of the outlet.

Feature of swirling flows in a centrifugal chamber is non-uniform distribution of the velocity field, which has three components: the rotational, axial and expendable or radial. In any swirling flow two zones with different distributions of the rotational speed of the radius can be marked out: a peripheral zone with the dependence of the rotational speed on the radius close to 1/r (so-called perfect swirl) and a central zone with a linear dependence of the rotational speed on the radius (rigid rotation). The dependence of the radius on the axial velocity in different sections can have, depending on ambient conditions, a complex form. Return flow, the precession of the axis swirl, etc. can exist.

All variety of flows does not fit the scheme proposed by the classics of swirled flows. Therefore, the current interest in the swirling flow doesn't abate. A large number of works are dedicated to numerical modeling and experimental investigation of dynamic processes in the swirl chamber.

Corresponding Author: I.A. Kuksov, Siberian State Industrial University (SibSIU), Novokuznetsk, Kirova St, 42, Russia.



Pic. 1: Chamber model: a - model of the experimental swirl combustion chamber, b - fragment of the computational grid, c - configuration version of industrial combustion chamber

The aim of this work is to define a three-dimensional velocity field at specified mass flow rate through the headers and pressure difference between upper and lower sections of swirl chamber.

Two of the most common types of fuel in the swirl chambers are pulverized coal fuel and coal-water slurry (CWS) fuel, injected under pressure into the chamber. An important factor to consider while designing power plants, based on such fuel, is the control of the air pollutant emission into the environment (mainly, CO and NO_x).

Important parameters of pulverized coal and CWS combustion, such as the degree of fuel and oxidizer mixing, the residence time of fuel particles in the combustion zone, the amount of pollutant emission can be determined by the analysis of gas-dynamic processes in the swirl chamber.

For burning the coal-water slurries on the site of Siberian State Industrial University (Novokuznetsk, Russia) was created swirl adiabatic chamber consisting of a cylindrical container and 8 four-row nozzles. (Pic. 1, a). It is designed for testing technologies that will be used to create automated power generating systems, running on fuel produced from coal enrichment wastes.

Fuel injection goes into this chamber through four separate injectors. Compressed air goes together with fuel through these injectors, thanks to that spraying of the fuel particles inside the swirl core is carried out. In the experimental set injectors, as well as air nozzles, are located in the side walls of the combustion chamber.

In the developed power plants the same principle of the tangential air and fuel supply will be used. Designed industrial complexes can have not cylindrical, but polyhedral form (Pic. 1,c), but as further calculations have shown it has no significant effect on the core part of the swirl.

The results of the analysis of gas-dynamic processes, combustion and characteristics of NO_x emissions in a similar designed combustion chamber by numerical simulation are given in [3]. In this article the analysis of numerical modeling allowed the authors to propose a method of reducing NO_x emission by choosing the optimal amount of air supply into the air reaction zone.

Due to the swirl in the swirl combustion chambers recirculation zone is created. Thanks to it there are stable combustion conditions due to reagents supply to the reaction area and their mixing. Valera-Medina, Syred and others revealed on the basis of experimental data analysis that the combustion of partially mixed mixture has a bigger influence on the formation of the central recirculation zone than the diffusion combustion [4].

The chamber, considered in this article is assigned to study the combustion processes, which occurs in partially mixed mixture mode formed by centrifugal injector.

Akhmetov and Shkadov give description of particle dynamics in the swirl combustion chamber [5]. The authors note that centrifugal force and viscous force (Stokes force) have the biggest influence on particle in the swirling flows. This fact must be considered designing the swirl combustion chambers.

MATERIALS AND METHODS

The air is supplied to the header with mass flow rate varying from 600 to 900 m^3/hour. With the diameter of header nozzle d=29 mm the speed at the section of supply canal is 40m/sec and Reynolds and Mach numbers are $Re=10^4$ and M=0,13. That means that turbulent subsonic jets flow out of header canals.

The amount of air supplied to the chamber was chosen in the way to provide a relatively low intensity of the swirl. As the earlier studies have shown, it provides a stable flame formation and less pollutant emission during fuel combustion in comparison with high-speed swirl [6, 7].

In the calculation, presented in this article, 2 of 4 compressed air and fuel injectors were used to fit the carried out experiments. These injectors are located at level of lower air inlets. Compressed air was supplied at the pressure of 3 atm. Fuel mass flow rate injected through the point injector was 0.013 kg/s.

Geometrically, air nozzles and fuel injectors are located in pairs in the opposite vertices of the inscribed rectangle.

Calculation of gas-dynamic processes in swirl chamber is carried out with software STAR-CCM+ (version 6.02), developed by CD-adapco. STAR-CCM+ is used to study swirl structures in the swirl combustion chambers. In [8] the calculation in STAR-CCM+ for similar chamber is described, obtained result coincides with the experimental data.

To describe the gas flow Euler models were used. Simulation of fuel particles was carried out by the Lagrangian phase.

Non-stationary Reynolds-Averaged Numerical Simulations (RANS) are used in calculations. This approach was chosen because of its bigger calculating efficiency with fair accuracy for a particular task in comparison with direct numerical simulation (DNS) and Large Eddy Simulation (LES).

In order to receive reliable results by the swirl processes simulation it is necessary to choose the most appropriate model of turbulence. As Hreiz and others pointed out in their paper, in similar conditions, among the models using Reynolds-Averaged Numerical Simulations, the Realizable k - ε model [9] demonstrates the best results in predicting the gas-dynamic parameters, such as the mean axial velocity.

This model is applicable for modeling systems with recirculating flows. In addition, Realizable k - ε model is suitable for modeling gas-dynamic processes with the processes of heat transfer.

The problem was solved in three-dimensional formulation. The model was built in full size to fit the existing experimental setup. The method of finite volume is used for the numerical calculation of the given mathematical problem. The time-step for the given calculation was chosen to be 0.01 sec. Computing method is implicit with the number of inner iterations set to 20.

The parameters of computational mesh are the following: polyhedral mesh, basic size of the cell - 0.005m, number of wall prismatic layers - 4, stretching coefficient of wall layers -1.5; number of volume mesh cells- 1 272 732. The mesh splitting with ratio of 0.05 was carried out in the area of the inlets. Besides nozzles and fuel injectors, mesh splitting is implemented in the interior of the chamber near the inlets of these channels. A similar mesh splitting for analogous problem is described in [10].

In addition, on the inlets and outlets boundaries the normal extrusion of the volume mesh was implemented in order to receive accurate results in these areas. In the extrusion areas the orthogonal cell layers were used.

The constructed volumetric mesh with the set parameters is sufficient to achieve calculation convergence. The fragment of mesh splitting is in picture 1b, where prismatic layers nearby inside surface of the supply channel are presented.

Boundary conditions are set according to the parameters of swirl chamber. On the inlet boundaries the mass flow rate and direction are set. The angle between the velocity direction and the inlet normal is 75 degrees. For the outlet boundary located on the top of chamber the outer pressure is specified. The x-axis x directed along vertical axis of the chamber (Pic. 1a) and axes y, z lay in perpendicular plane to x.

To estimate the gas-dynamics while fuel combustion, gas temperature was set to 1100°C, which is equal to the volume averaged temperature in the combustion chamber during the experiments.

For the turbulence model on the inlet boundaries the following initial values were set. The intensity of the turbulence is assumed to be 0.01, the turbulent viscosity ratio is assumed to be 10.

For the wall area High y+ Wall Treatment model was used. This model assumes that the closest to the wall cell is in the logarithmic area of the boundary layer.

For the calculation Coupled Flow model was used, in which the mass and moment equations are computed simultaneously. The advantage of this approach is the independence of the number of iterations required to achieve convergence, from the mesh size.

In this calculation only the gas-dynamic processes were simulated, so the combustion models were not activated.

For the control of task convergence the analysis of nonviscous physical parameters and comparison of mass flow rate on inlet and outlet boundaries was used.

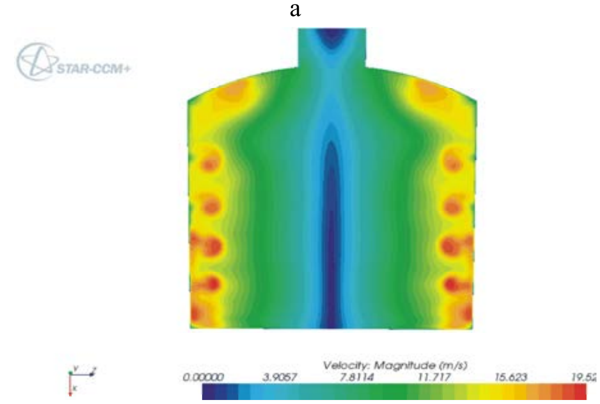
To avoid the problem of divergence of turbulence model at the initial stages of the calculation that may happens because of the large pressure drop when the gas flow is simulated in the input channels, the target mass flow was increased gradually.

The influence of design and operation parameters of the injector, through which a stream of air injects the fuel, on the gas flow in the chamber is described in [11]. Therefore, to obtain accurate numerical solutions while creating the model it is important to specify the characteristics of the injector. There are two methods to specify the inlet stream: substance flow modeling inside the nozzle and definition of flow at the inlet on the basis of empirical data of the nozzle work. In the calculation, described in this article, the second method is used.

RESULTS AND DISCUSSIONS

In Picture 2 velocity distributions in section passing through an axis x are presented. In Picture 2a projections of a velocity vector field to a plane of section are represented. Velocity vectors perpendicular to the planes of section are represented by points and sizes of the represented vectors depend on angle of a slope and the velocity magnitude. From this picture it follows, that from the middle of the chamber there is an axial flow along axis x. It is caused by the pressure difference, set in

a



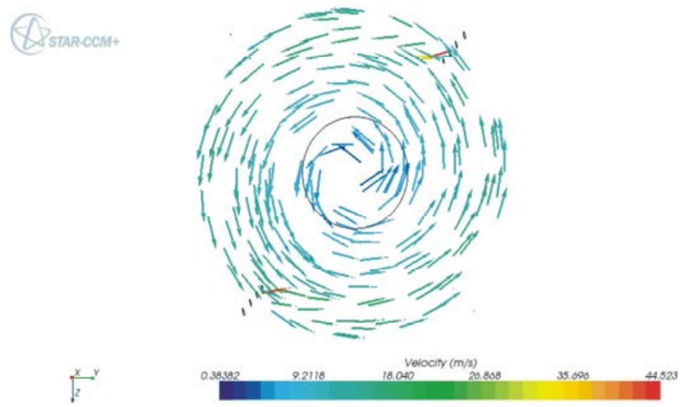
b

Pic. 2: Velocity field in the axial section: a - projection component of the velocity field in the axial section, b - isolines of velocity magnitude

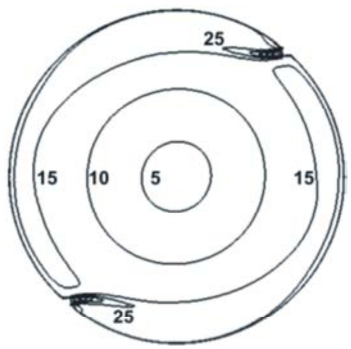
outlet section. In Picture 2b isolines of velocity magnitude are presented. Its upper-range values are reached on a course of streams.

The picture shows that the configuration of the flows from the nozzles is stored in this section. In addition, the areas of elevated rates are forming under the arch of the combustion chamber. It is assumed that the presence of a number of successive toroid-like flows increase the residence time of the fuel particles in the reaction zone.

In Picture 3 velocity distribution in planes *y, z* are presented. In picture 3a the velocity vector field is presented (the size of vector doesn't correspond to the velocity magnitude). The notion of the size of a velocity vector can be gained from Pic. 3b.

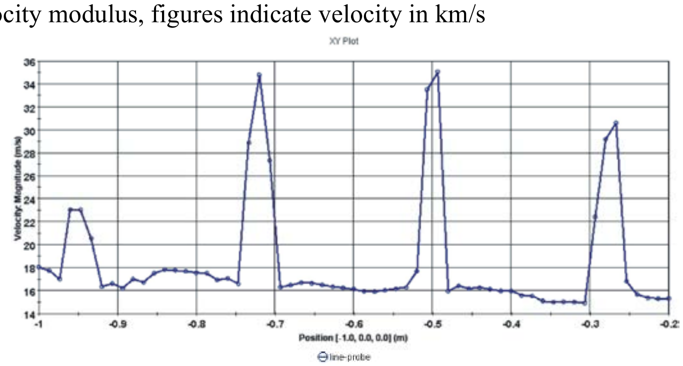


a

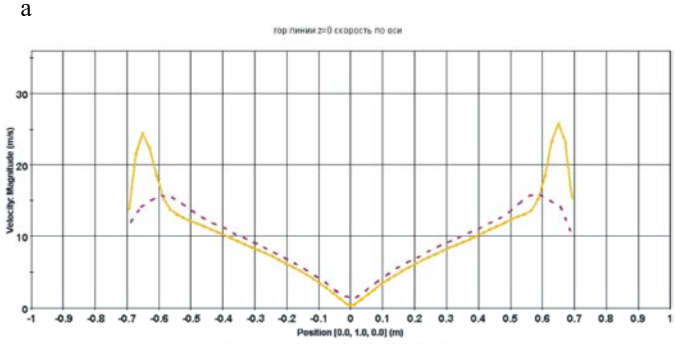


b

Pic. 3: Velocity field in the cross-section, perpendicular to axis: a- vector diagram of the directions of velocity vector, b- isolines of velocity modulus, figures indicate velocity in km/s



a



b

Pic. 4: Plots of the velocity magnitude: a - on a vertical line along at a distance of 0.1 m from air inlets, b – on a horizontal lines along at the level of lower and upper nozzles

According to the above data the flow in the swirl chamber can be divided into two sections. In the first section $0<r<R_1$ the flow is analogous to Couette - Poiseuille flow with rotating cylinder having angular rate Ω, radius R_1 and average pressure drop in the upper and lower sections of cylindrical chamber. In the second section, adjacent to the edge of cylinder, complex flow is formed due to interaction of two steams and boundary layers.

In the Picture 4 there are velocity-time diagrams: on vertical line along at a distance of 0.1 m from air inlets (a) and on horizontal lines along at the levels of lower and upper nozzles.

Pictures 2, b and 4 show that fuel injectors influence the flows from the lower air nozzles.

CONCLUSIONS

Based on the obtained results the optimal characteristics of the fuel injection can be estimated to ensure the most efficient modes of combustion, for example, the nozzles location and mass flow rate of compressed gas supplied with the fuel.

The presented calculations allowed to define complex three-dimensional flow in the vortex chamber as a flow superposition of Couette-Poiseuille flow with radial gas consumption and flow between two cylinders with gas blowing through collector nozzles (the external cylinder) and gas withdrawal through internal cylinder.

ACKNOWLEDGEMENT

Work is carried out within the project No. 2010-218-02-174 by the governmental order of the Russian Federation of April 9, 2010 No. 218, “About state support measures, about progress of cooperation of the Russian higher educational institutions and organizations, realizing complex projects on development of high-technology manufacturing”.

REFERENCES

1. Syred, N. and J.M. Beér, 1974. Combustion in swirling flows: A review. Combustion and Flame, 23(2): 143-201.
2. Vatistas, G.H., S. Lin and C.K. Kwok, 1986. Theoretical and experimental studies on vortex chamber flows. AIAA Journal, 24(4): 635-642.
3. Choi, C.R. and C.N. Kim, 2009. Numerical investigation on the flow, combustion and NOx emission characteristics in a 500 MWe tangentially fired pulverized-coal boiler. Fuel, 88(9): 1720-1731.
4. Valera-Medina, A., N. Syred, P. Kay and A. Griffiths, 2011. Central recirculation zone analysis in an unconfined tangential swirl burner with varying degrees of premixing. Experiments in Fluids, 50(6): 1611-1623.
5. Akhmetov, V.K. and V.Y.A. Shkadov, 2009. Numerical modelling of viscous vortical flows for technical appendices. Publishing House ASV, pp: 176.
6. Plessing, T., C. Kortschik, N. Peters, M.S. Mansour and R.K. Cheng, 2000. Measurements of the turbulent burning velocity and the structure of premixed flames on a low-swirl burner. Proceedings of the Combustion Institute, 28(1): 359-366.
7. Cheng, R.K., D.T. Yegian, M.M. Miyasato, G.S. Samuelsen, C.E. Benson, R. Pellizzari and P. Loftus, 2000. Scaling and development of low-swirl burners for low-emission furnaces and boilers. Proceedings of the Combustion Institute, 28: 1305-1313.
8. Anokhina, E.S., 2012. Modelling of stationary vortical structures in the model combustion chamber. In the Proceedings of 18th All-Russia scientific conference of students-physicists and young scientists, Krasnoyarsk, pp: 622-623.
9. Hreiz, R., C. Gentric and N. Midoux, 2011. Numerical investigation of swirling flow in cylindrical cyclones. Experiments in Fluids, 50(6): 1611-1623.
10. Butyrev, A.E., M.P. Galanin, V.G. Gnedenko, A.V. Pereslavtsev and S.S. Tresvyatskiy, 2007. Mathematical modelling of gas flow in vortical chambers with tangential blowing. Preprint of Institute for Applied Mathematics, 85: 1-27.
11. Yin, H. and R. Dai, 2011. Experimental Study on the Non-reacting Flowfield of a Low Swirl Burner. Energy and Environment Research, 1(1): 105-110.