Numerical modeling of low-rank coals combustion with different moisture content at power plants of kazakhstan

Askarova A.S., Bekmukhamet A., Bolegenova S.A., Maximov V., Beketayeva M.T., Ospanova Sh.S.

***Abstract*—** In this paper the results obtained by the method of numerical modeling of Ekibastuz coal burning in furnace of Kazakhstan Power Plant. Numerical experiment was carried out on the basis of three-dimensional equations of convective heat and mass transfer, taking into account the heat propagation, heat radiation, chemical reactions and multiphase structure of the medium to predict the influence of different water content in coal on overall furnace operation and formation of combustion products.

***Keywords*—** Combustion, pulverized coal, coal-dust flame, turbulence, carbon monoxide, thermal power station, heat and mass transfer, multiphase structure, hydrocarbonaceous, carbonic oxide.

I. Introduction

THE power system of Kazakhstan is presented mostly by thermal power stations more than 70 % of total amount of the electric power are generated. Thermal power stations work on solid fuel when pulverized coal combustion is predominant. Also natural gas and black oil can be used. Tendency of low grade coals usage to increase in domestic industry, it becomes especially important to develop and introduce new energy-saving technologies of solid fuel consumption and reduction in pollutant substance emissions. Applied technology of direct burning of low-grade coals in furnaces does not provide demanded reliability of working equipment and protection of environment from harmful combustion products due to approximately 50 % of pollutant emissions from stationary sources comes from thermal power enterprises and about 33 % are emitted by ferrios and non-ferrious metallurgy and mining.

Industrial implementation of any new technology is not possible without preliminary analysis of advantages and disadvantages suggested method. The rapid development in computer sciences gives the advance to computational techniques to be used for simulation of complex combustion processes in industrial furnaces.

Products of combustion contain different harmful substances and the emission of this components grows in to a great problem. Industrial development causes an increase in hydrocarbonaceous fuels' consumption. These fuels contain harmful and poisonous components such as carbonic oxide (CO), nitric oxide (NO), sulphur dioxide, acid sulphate, lead combinations and different hydrocarbons etc.

To decrease emissions of harmful substances various methods are applied, including special fire regimes (organization of combustion process), which suppresses the formation of harmful substances in flame and two-stage burning, when the burners work with low air surplus. In this way numerical experiments became one of the most effective and suitable means for detail analysis and in-depth study of physical and chemical phenomena.

In contrast to construction of an operating reduced model of the chamber, the three-dimensional modeling with the application of modern computer technology enables to carry out deep analysis of all chamber's parameters and save time and finances. At the same time, without additional inputs, it is possible to obtain the full set of characteristics of a convective heat and mass transfer process in reactive media, intervene flexibly in the process at any stage and reproduce separate technical solutions (the configuration of the fire chamber and assembly and construction burners), to model the formation of harmful dust and gaseous emissions and to investigate the influence of previous preparation of coal on its ignition and combustion stabilization.

II. Mathematical model

Combustion of coal-dust flame is a very complex physical-chemical process for mathematical analysis. In the boilers, where coal-dust fuel burns down in the air stream, many interrelated process occur. They are: complex aerodynamics, combustion under conditions of constantly changing temperature and component concentrations, heat exchange between the flame and the chamber surface. Simultaneous formation of carbon, nitrogen and sulphur oxide, corrosion hazardous and carcinogenic substances occurs.

A.S. Askarova[1]. Al-Farabi Kazakh National University, Almaty, 050038 Kazakhstan (e-mail: Aliya.Askarova@kaznu.kz).

A. Bekmukhamet[2] , He is now 1st course PhD student of the Departament of Thermal Physics and Technical physics, Al-Farabi Kazakh National University, 050038 Kazakhstan (corresponding author to provide phone:+7-701-312-68-38 ; fax: no ; e-mail: Kazsat2006@gmail.com).

S.A. Bolegenova[3], She is now 3rd course PhD student of the Departament of Thermal Physics and Technical physics, Al-Farabi Kazakh National University, 050038 Kazakhstan (e-mail: Symbat.80@mail.ru).

V.Yu. Maximov[4] He He is now 2nd course PhD student of the Departament of Thermal Physics and Technical physics, Al-Farabi Kazakh National University, 050038 Kazakhstan (e-mail: maximov.v@mail.ru).

A numerical experiment was carried out on the basis of three-dimensional equations of convective heat and mass transfer, taking into account the heat propagation, heat radiation, chemical reactions and the multiphase structure of the medium. To describe 3-D motion of reacting flows in chambers, the following set of differential equations was used [1]:

For conclusion of balancing ratios of the stationary control element of volume or control element of weight (Figure 1). It is assumed that the center of gravity of the selected element moves with the velocity of flow. This corresponds to a stationary control volume sound approach for the Euler's flow. Change the value of the transport is described in a single fluid element. Value of transport size is defined in each point of considered area.

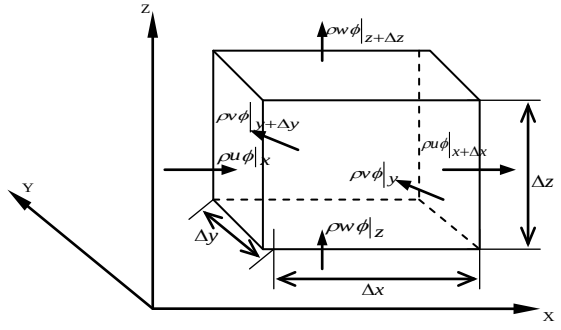

Fig 1. Control volume for the generalized transport equation

When converting from a finite limit to the infinitesimal volume element is obtained by controlling the differential equation describing the conservation of the transport variable ϕ:

$$\frac{\partial(\rho\phi)}{\partial t} = -\frac{\partial(\rho u_1\phi)}{\partial x_1} - \frac{\partial(\rho u_2\phi)}{\partial x_2} - \frac{\partial(\rho u_3\phi)}{\partial x_3} + \frac{\partial}{\partial x_1}\left[\Gamma_\phi \frac{\partial\phi}{\partial x_1}\right] + \quad (1)$$

$$+ \frac{\partial}{\partial x_2}\left[\Gamma_\phi \frac{\partial\phi}{\partial x_2}\right] + \frac{\partial}{\partial x_3}\left[\Gamma_\phi \frac{\partial\phi}{\partial x_3}\right] + S_\phi$$

Replacing in equation (1) the convective and diffusive transport of flux density, cross-border control volume, we obtain:

Density of the convective flow: $\Phi_{(K),\,j} = \rho u_j \phi$

Density of diffusion flux: $\Phi_{(D),\,j} = \Gamma_\phi \frac{\partial\phi}{\partial x_j}$

Then, based on the data relationships (1) can be written as:

$$\frac{\partial(\rho\phi)}{\partial t} = -\frac{\partial\Phi_{(K),\,j}}{\partial x_j} + \frac{\partial\Phi_{(D),\,j}}{\partial x_j} + S_\phi \quad (2)$$

We write equation (2) in vector form:

$$\frac{\partial(\rho\phi)}{\partial t} = div\left(-\rho\vec{u}\phi) + \Gamma_\phi grad\phi\right) + S_\phi \quad (3)$$

And in tensor form, equation (2) becomes:

$$\frac{\partial(\rho\phi)}{\partial t} = -\frac{\partial\left(\rho u_j\phi\right)}{\partial x_j} + \frac{\partial}{\partial x_j}\left[\Gamma_\phi \frac{\partial\phi}{\partial x_j}\right] + S_\phi \quad (4)$$

Here the index j characterizes the spatial direction component of the velocity u_j in the chosen Cartesian coordinate system with coordinates x_j, where j = 1, 2, 3.

Then , we can use: $x_1 \equiv x; \quad x_2 \equiv y; \quad x_3 \equiv z$

$$u_1 \equiv u; \quad u_2 \equiv v; \quad u_3 \equiv w \quad (5)$$

Value of ϕ change by time : $\frac{\partial(\rho\phi)}{\partial t}$

ϕ change due to convective transport: $\frac{\partial\left(\rho u_j\phi\right)}{\partial x_j}$

ϕ change due to molecular exchange phenomena : (Diffusive exchange). $\frac{\partial}{\partial x_j}\left[\Gamma_\phi \frac{\partial\phi}{\partial x_j}\right]$

The source (sink) term for the quantity of ϕ: s_ϕ

A standard k-ε model of turbulence has been used to close off the system and model turbulent viscosity [2,3]:

$$\frac{\partial}{\partial\tau}(\bar{\rho}\bar{k}) = -\frac{\partial}{\partial x_i}\left(\overline{\rho u_i}\bar{k}\right) + \frac{\partial}{\partial x_i}\left[\frac{\mu_{eff}}{\sigma_k}\frac{\partial k}{\partial x_i}\right] + P - \bar{\rho}\varepsilon \quad (6)$$

$$\frac{\partial}{\partial\tau}(\bar{\rho}\bar{\varepsilon}) = -\frac{\partial}{\partial x_i}\left(\overline{\rho u_i}\bar{\varepsilon}\right) + \frac{\partial}{\partial x_i}\left[\frac{\mu_{eff}}{\sigma_\varepsilon}\frac{\partial\varepsilon}{\partial x_i}\right] + c_{\varepsilon 1}\frac{\varepsilon}{k}P - c_{\varepsilon 2}\bar{\rho}\frac{\varepsilon^2}{k}$$

Where

$$\Pi = \left[\mu_t\left(\frac{\partial\bar{u}_i}{\partial x_j} + \frac{\partial\bar{u}_j}{\partial x_i}\right) - \frac{2}{3}\bar{\rho}k\delta_{i,j}\right]\frac{\partial\bar{u}_i}{\partial x_j} \quad (7)$$

For turbulent viscosity: $\mu_t = c_\mu \rho k^2/\varepsilon$.

Calculation of turbulent flows with chemical reactions is based on the knowledge of chemical kinetics and modeling methods of turbulent transfer processes. To determine the source term in equation for componenets which is related to chemical reaction rate it's necessary to model correctly kinetics of chemical reactions. These reactions take place between fuel and oxidant in combustion space of the boiler. Local distribution of reacting components and temperature significantly influence on the reaction rates. The amount of energy, emitted in chemical reactions, the flame temperature and the nature of combustion products are very important characteristics in estimation of solid substances influence. Using the solid particles as fuel is most probable.

The coal combustion model is divided into five submodels for drying, pyrolysis (release of volatiles), combustion of volatiles, carbon monoxide and residual char. Coal devolatilization is considered to run in a single step reaction forming a general hydrocarbon CxHy. The amount and composition of the hydrocarbon species have been taken from elementary and ultimate analysis of the coal. The model of coal dust combustion which is being used in this paper takes into account an integral oxidizing reaction of fuel components down to a stable final products of reaction. Intermediate reactions and the formation of intermediate products haven't

been taken into consideration. The model of integral reaction was used based on according to the fact that many chemical reactions proceed several stages. The slowest reaction determines the rate of the whole reaction.

The combustion of volatiles in the gas phase is assumed to be controlled by turbulent mixing, using the Eddy Dissipation Concept. The Eddy Dissipation Model according to Magnusson et al. is used to predict the combustion of the volatiles and the carbon monoxide formed during char combustion. Gaseous fuels are treated like volatiles. The char reaction rate is governed by the rate of oxygen diffusion to the surface and the kinetic rate of chemical reaction at the surface.

Simplified models are being used in this paper. These models take into account only chemical reactions of key components since the detail modeling of all passing reactions is possible only in the simple cases, because of large computational costs. For example, in the case of carbon oxide burning,. Information about reactions and their kinetic data have been taken from [4].

Within the combustion of fossil fuel, nitric oxide is produced by different reaction paths. The main reactions are the oxidation of the molecular nitrogen in the air (thermal NO) and the oxidation of the fuel bounded nitrogen (fuel-NO). To describe the rates of forming and reducing nitric oxide during combustion global reaction schemes proposed by Mitchell and Tarbel [5] are used.

In this paper software package FLOREAN [1, 6-9] for 3-D modeling of coal-dust combustion in furnaces of real-sized boilers was used. This program enables to calculate velocity components *u*, v, *w*, temperature *T*, pressure *P*, concentration of combustion products and other characteristics of combustion process all over the combustion space and at its exit. Pressure is determined through the connection between the continuity equation and the equation of motion by means of Patankar's Simple-method [10].

III. Results of CFD Studies

In present work CFD simulation tool was used to simulate and analyze low-grade coal combustion and flow properties of industrial furnaces of Kazakhstan Power Plants. The simulations of the furnaces of steam generators placed at Almaty Power Plant have been carried to predict the influence of different water content in coal on overall furnace operation and formation of combustion products.

Fig. 2 shows temperature distribution along the chamber height for burning of coal with different water content in initial air-fuel stream. The experimental data [11] are given here for coal burning with 7% moisture. Curve minimums caused by low temperature of fuel mixture supplied in the chamber space through the burners. Differences of calculated and experimental values of temperature are seen in the region of ignition and extinction. Figure 3 shows difference in temperature fields for different initial water content in burning coal. We can see that higher water content causes decrease in temperature in this section of combustion chamber.

Figures 4-10 show picture of fuel burning with the fields of Oxygen, Carbon Oxide CO and Carbon Dioxide CO_2 and Nitrogen Oxide NO concentration. The main formation regions of gases in the coal dust torch take place at the level of burners in the central part of the furnace volume.

The maximum difference of calculation and experiment (fig. 9) is seen in the region of ignition and extinction of flying matters. The main reason for this is that there is a determination of the gas burning rate by one stage model of piroliz where the set of kinetic constants for all the entire temperature range are used. The figures show that the regions with maximum difference coincide with regions having a high concentrations of carbon oxide.

At the chamber furnace outlet we can see that decrease in coal moisture results in CO_2 decrease, (fig. 9) and simultaneous CO increase (fig. 7-8). Water content (fig.6) and oxygen concentration (3,4) in the total gas mixture at the outlet consequently increase with water content in coal (fig. 4-5). Same development is seen in the whole furnace volume (fig. 4,6). Difference in NO formation for studied coal content is seen mostly in the region of burners location. At the furnace outlet the difference in NO concentrations is slight appearing (fig.10). The distribution character of all concentrations, listed in this paper is modeled sufficiently and it agrees with the experimental data [11].

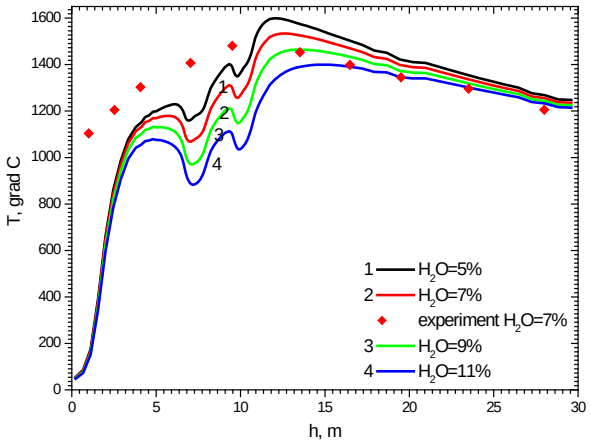

Fig. 2 Temperature distribution along the chamber height for coal burning with different water content.

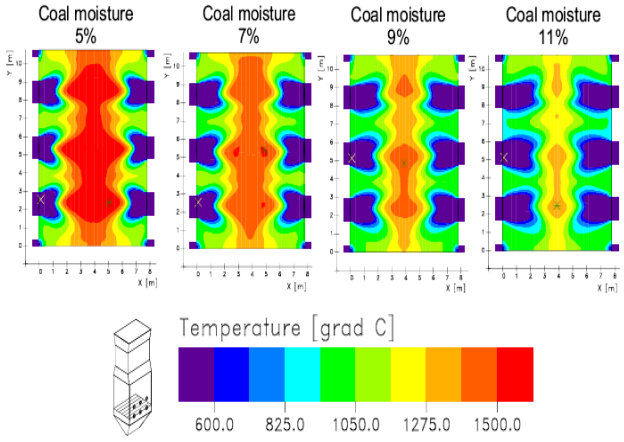

Fig. 3 Temperature distribution in cross section of the furnace in the lower level of the burners for coal burning with different water content.

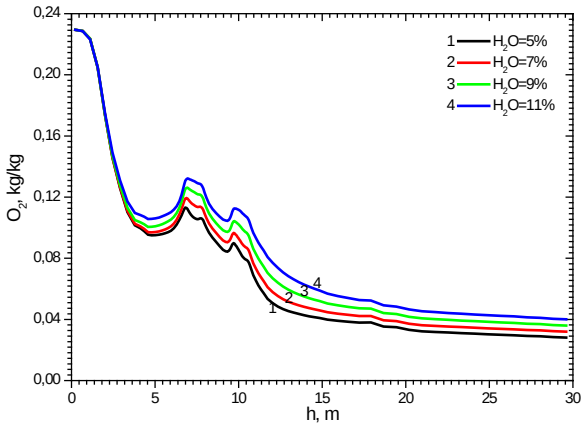

Fig. 4 O_2 concentration distribution along the chamber height for coal burning with different water content.

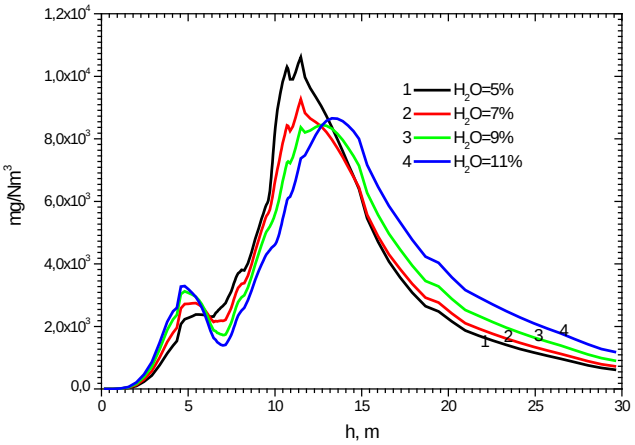

Fig. 7 CO concentration distribution along the chamber height for coal burning with different water content.

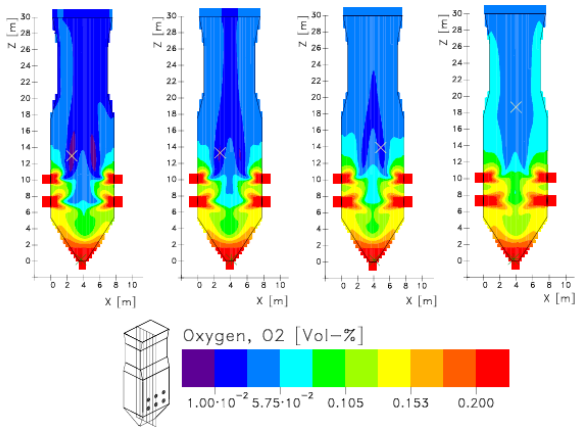

Fig. 5 Oxygen concentration distribution in the central section of the furnace for coal burning with different water content.

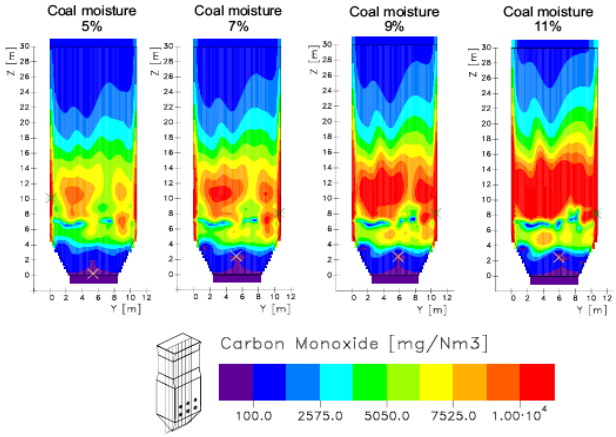

Fig. 8 CO distribution in the central section of the furnace for coal burning with different water content.

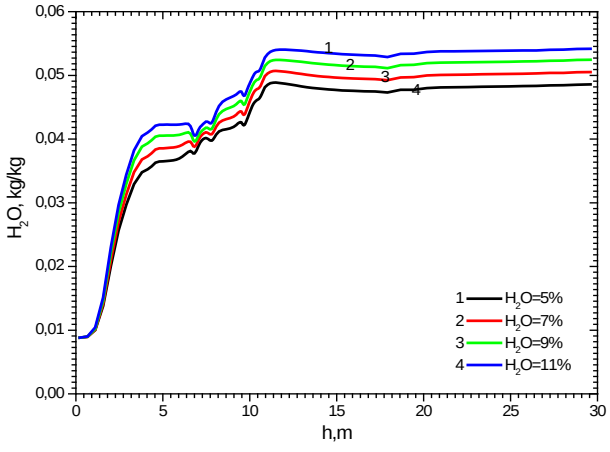

Fig. 6 Water concentration distribution along the chamber height for coal burning with different water content.

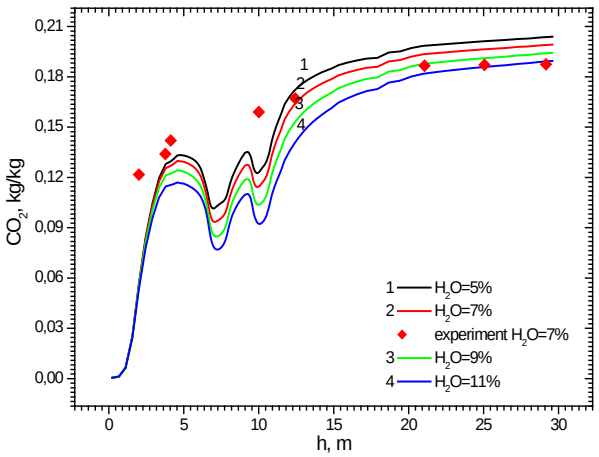

Fig. 9 CO_2 concentration distribution along the chamber height for coal burning with different water content.

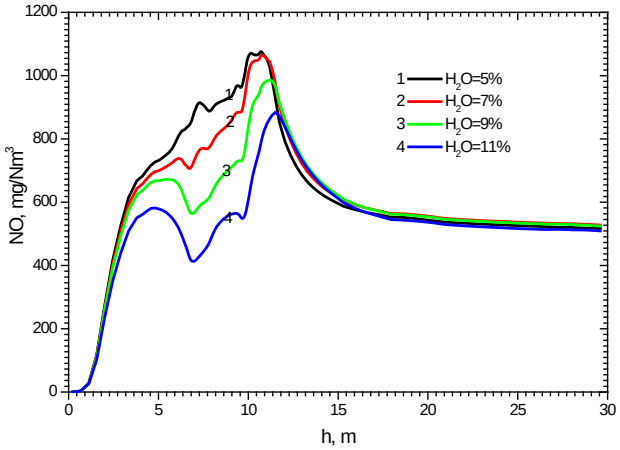

Fig. 10 NO concentration distribution along the chamber height for coal burning with different water content.

IV. Conclusion

In present work CFD simulation tool was used to numerically simulate and analyze low-grade coal combustion and flow properties of industrial furnaces of Kazakhstan Power Plants. The simulations of the furnaces of steam generator of Almaty Power Plant have been carried to predict the influence of different water content in coal on overall furnace operation and formation of combustion products. Results presented in the way of 3D fields of temperatures, velocities and concentrations of the combustion products CO, CO_2, NO_x etc.

CFD prediction of combustion process of combustion chamber can be very useful for engineers to choose an appropriate operating method to successful overall combustion optimization in furnaces to improve energy efficiency and control pollutant emissions.

V. References

[1] Müller, H. Numerische Berechnung Dreidimensionaler Turbulenter Stromungen in Dampferzeugern mit Wärmeübergang und Chemischen Reaktionen am Beispiel des SNCR-Verfahrens und der Kohleverbrennung. *Fortschr.-Ber. VDI-Verlag.* Vol. 6, p 158, 1992.

[2] Launder, B. E. et. al. Lectures in mathematical models of turbulence. Academic Press, London, 1972.

[3] Launder B., Spalding D. The numerical computation of turbulent flows // Comp. Meths. Appl. Mech. Eng, 1974.

[4] Lockwood, F. G. et. al. An improved flux model for the calculation of radiation heat transfer in combustion chambers. ASME Paper 76–HT–55, 1976.

[5] Mitchell, J. W., Tarbell, J. M. A kinetic model of nitric oxide formation during pulverized coal combustion, AIChE Journal, Vol. 28, 1982, S.311.

[6] Leithner, R., Müller, H. CFD studies for boilers // *Second M.I.T. Conference on computational Fluid and Solid Mechanics*, Cambridge, 2003.

[7] Askarova A.S, Karpenko EI, Lavrishcheva Y.I, Messerle V.E, Ustimenko A.B Conference Information: 3rd International Workshop and Exhibition on Plasma-Assisted Combustion // IEEE TRANSACTIONS ON PLASMA SCIENCE**,** Volume: 35, Issue**:** 6, Pages**:**1607-1616, 2007.

[8] Askarova A.S, Messerle V.E, Ustimenko A.B, Nagibin A. Pulverized coal torch combustion in a furnace with plasma-coal system //THERMOPHYSICS AND AEROMECHANICS Volume17, Issue3, p 435-444, 2010.

[9] Askarova AS, Karpenko EI, Messerle VE, Ustimenko AB. Mathematical modelling of the processes of solid fuel ignition and combustion at combustors of the power boilers Conference Information: 7th International Fall Seminar on Propellants, Explosives and Pyrotechnics, CHINA // THEORY AND PRACTICE OF ENERGETIC MATERIALS, VOL VII Pages: 672-683, 2007.

[10] S.V. Patankar, Numerical Heat Transfer and Fluid Flow, Hemisphere Publ. Corp., N. Y., 1980.

[11] Aliarov B.K., Ustimenko B.P., Buhman M.A. // Report № 01910010550. Kaz. SRIETP. Development and inculcation vortical three-channels burners on the boilers P-39-2 Ermnakovskaja Power Plant, Almaty, 59 p., 1991.

Improvement of Aerodynamics Characteristic of Heavy Trucks

Dr. Imad Shukri Ali and Aws Akram Mahmood

***Abstract*--** The present work study experimentally and theoretically the effect of changing the aerodynamic shape of a tractor trailer scaled model on its aerodynamic drag and improving its aerodynamic characteristic by using modification added on the trailer body .The experiment work was executed by using wind tunnel with Reynolds No. equals to 10^6 based on the truck length on model of $1/30^{th}$ scale of (Mercedes - Benz 1844 ACTROS Container- Trailer) .The modifications added to rear was base flap and base flap with splitter, using four inclined angle (0˚,10˚,20˚and30˚).The experimental results shows that there is a good reduction on aerodynamic drag by approximately equals to 18% when the inclined angle equals 10˚.The splitter addition produced an improve in aerodynamic reduction on the behavior of the base flap at β=20˚ with 21% drag reduction.

The theoretical study was executed by using CFD package which is ANSYS FLOTRAN 12.1 by using CAD preparation of two dimensional model with zero yaw angle .The results obtained for x-component velocity , velocity vectors and stream lines shows good agreements between the experiment and theoretical work.

***Keywords*--** Drag reduction, CFD , Turbulent flow , Tractor trailer

Nomenclature

Symbl	***Meaning***	***Units***
$\Delta C_P\%$	*Pressure coefficient percentage change*	--
$\Delta C_D\%$	*Drag coefficient percentage change*	---
$\Delta\mu(V_t)$	*Fuel saving on the drag coefficient change*	*Litter*
$C_D(V_t)$	*drag coefficient depending on velocity of vehicle*	---
$\Delta C_D(V_t)$	*Drag coefficient % change depending on velocity of vehicle*	---
Δh_{water}	*Water reading head from manometer*	*m*
V_t	*`velocity of vehicle*	km/hr
P_{fb}	*Average fore-body pressure*	N/m^2
P_b	*Average base pressure*	N/m^2
P_n	*Local (tap) pressure*	N/m^2
P_∞	*Free stream pressure*	N/m^2
P_d	*Dynamic pressure*	N/m^2
R_e	*Reynolds number*	--
θ	*Yaw angle*	*deg.*
β	*Base flap modification inclined angle*	*deg*
ρ	*Density*	kg/m^3
τ	*Shear wall*	N/m^2

Dr. Imad Shukri Ali and Aws Akram Mahmood are with College of Eng., Mech, Eng.Dept, Babylon University- Hilla, Iraq. Email: emad.Ali6@gmail.com; aoss.akram@hotmail.com.

I. Introduction

ONE of the most important aim of the aerodynamic drag reduction researches is to save energy and to protect the global environment, fuel consumption reduction is primary concern of automotive development. In vehicle body development, reduction of drag is essential for improving fuel consumption and driving performance, and also the aerodynamically refined body is aesthetically attractive, it will contribute much to increase the vehicle's capacity.

The aerodynamic drag of heavy road vehicles participates significantly on the overall energy of vehicles efficiency and thus represents a challenge for its optimization. Although the aerodynamic development of passenger and sport cars has seen much growth, it can be same for commercial heavy vehicle. The aerodynamic design of heavy vehicle tractor-trailer still needs more development for minimum fuel consumption.

Heavy vehicle are aerodynamically inefficient compared to other ground vehicle due to their larger frontal area and bluff bodies shape. It is estimated that a 40-tons articulated truck travelling at 60 mph consumes about 34 Litters of fuel to overcome drag across a 100-mile high way stripe ; an average car would consume four time less under similar condition *Zulffaa,K.* et.al**,**2012[14]**..** More energy is consumed in overcoming aerodynamic drag as the vehicle speed up. Reducing fuel consumption for heavy vehicles can be achieved by altering truck shapes to decrease the aerodynamic resistance (drag). It is conceivable that present-day truck drag coefficients might be reduced by as much as 50%,this reduction in drag would represent approximately a 25%reduction in fuel use at highway speeds,an estimated total savings of $1.5 billion per year can be recognized in the United States alone for just a 6% reduction in fuel use .This reduction represents 1% of all fuel use in the united state Fred Browand , et.al ,2004[6].

Heavy tractor trailer geometry may have many aspects geometrical configuration that contributes to drag force and can be articulated as source of drag. These drag source can present as follows *Nabeh N.*,2010 [13]:

1-Tractor geometry.
2-Gap between the tractor and trailer.
3- Under body drag.
4- Wheel aerodynamic.
5- Rear base of trailer.
6- Side mirror.