한국에너지공학회(2000년도)
춘계 학술발표회 논문집 P43~50

Numerical Simulation of the Flow Field inside a New 1 Ton/Day Entrained-Flow Gasifier in KIER

LI Xiangyang, Choi Youngchan*, Park Taejun*
Tsinghua University, China
**Korea Institute of Energy Research (KIER)*

Abstract: *The flow field of a 1Ton/Day entrained-flow gasifier constructed in KIER was numerical simulated in this paper. The standard k-ε turbulence model and SIMPLE procedure was used with the Primitive-Variable methods during computation. In order to find the influence factors of the flow field which may have great effects on coal gasification process inside gasifier, different geometry parameters at various operating conditions were studied by simulation methods. The calculation results show that the basic shape of the flow field is still parabolic even the oxygen gas is injected from the off-axis position. There exist an obvious external recirculation zone with a length less than 1.0m and a small internal recirculation region nears the inlet part. The flow field inside the new gasifier is nearly similar as that of the old 0.5T/D gasifier at same position if the design of burner remains unchanged.*
Keywords: *numerical simulation, k-ε turbulence model, flow field, entrained-flow gasifier.*

1. Introduction

Coal gasification is a key process for the commercial scale Integrated-coal Gasification Combined Cycles (IGCC) power plant. Being one of the most competitive and hopeful coal gasification technologies, the slurry feed type entrained-flow coal gasifier has been carefully studied in KIER and gained many important achievements after many years research. Based on the practices and experiences accumulated on a 0.5T/D entrained-flow gasifier, a new designed 1T/D coal gasifier is under construction recently by the coal gasification application research team in KIER. In order to predict the characteristics of coal gasification inside this new reactor in advance as well as help to improve the current design, a comprehensive mathematics model of this entrained-flow new coal gasifier is currently being built in KIER. Being the first step and foundation of the complex simulation model for the coal gasification process, the flow field of this 1.0T/D new gasifier is numerically simulated in this paper.

In the past 30 years, numerical simulation method has attracted many attentions for its advantages of low cost, speed, reducing operation risk, helping in the optimum of the system design and so on. Therefore, numerical simulation method is regarded as a powerful tool to predict the complicated physical and chemical process except from highly effective experimental research work. With the rapid development of computer technology in recent years, numerical simulation of some complex flow field has been possible and numerous research works had been published [1]-[3]. At present, even some general codes or commercial software are available for simulate the common flow field. But due to the complicated nature of turbulence and different initial and boundary conditions, it still need special considerations and modifications for a certain engineering object if it need carefully and detailed research in mechanism level. In the previous modeling work in KIER, Liu X.J. et. al.[4] calculated the cold flow field of 0.5T/D gasifier by using the Vorticity-Stream Function method. Assuming the inlet coal slurry to be some kind of primary air with same momentum at cold conditions, the calculation result shows that owing to the high injecting velocity of second flow of oxygen, there is an obvious central recirculation zone near the burner.

In this work, the flow field inside the 1.0T/D new gasifier is predicted by using the primitive-variable approach. Meanwhile, the influence of the solid phase is not considered during this cold flow field calculation. In order to meet the needs for optimum the design of new reactor and burner, parameter study method was applied in this prediction work, because parameter studies can provide a better understanding of the reactor performance for various design conditions utilizing the model. Since in the computer program, it is easy to change the geometry parameters and operating conditions to see the effects of their influences quickly, thus it can greatly reduce the time and money comparing with the corresponding experimental investigation. Therefore, calculation of the flow filed inside the new gasifier at different design conditions can help people to know if the coal slurry particles can mix well or distribute better inside the new designed reactor than in that of the old gasifier.

2. Scheme of the KIER new coal gasifier

The scheme of the oxygen-blown, entrained flow coal gasifier in KIER is shown in Fig. 1. The coal-water-mixture (CWM) is injected into the gasifier through the center hole of the burner while the oxygen gas is blowing in through eight surrounded holes during gasification. In order to discrete the slurry into fine particles and small droplets, the oxygen gas is injected in with a certain angle at high speed (Fig. 2).

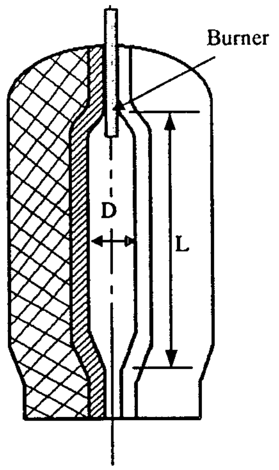


Fig.1 Scheme of the KIER gasifier

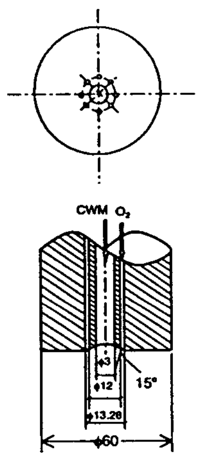


Fig.2 Detailed structure of the burner

In order to simplify the problem, the oxygen nozzle was assumed to be an annular one with the same diameter and the same flow rate as that of the eight surrounding holes during simulation. Therefore, this three-dimensional problem can be reduced to a two-dimensional axial-symmetrical case. The designed oxygen/coal ratio is about 0.8 for 1T/D new gasifier during operation, which means the oxygen feed rate will be 0.8Ton/Day. If changes to volumetric units, the oxygen gas flow in rate will be $Q=23.3(Nm^3/h)$. From the geometric data of the design, the oxygen inlet velocity will be $U_{in}=255m/s$. It can be seen that the oxygen blowing in velocity is very high through a very small inlet diameter ($D=1.28mm$). Meanwhile, the influence of coal slurry injection was not considered in this cold flow field simulation, therefore the center hole was assumed to be solid and the surface was treated as solid wall. But in real hot coal gasification process, the coal slurry was injected in through center hole at high speed. The solid phase will greatly influence the gas flow field and this effect must be taken into account.

3. Mathematics model

3.1 Governing Equations

According to the above description, the flow field inside the gasifier is axis-symmetrical referring to the centerline. If the cylindrical coordinates system is used, the 3-D problem can be reduced to a two-dimensional problem. Meanwhile, the flow field inside the gasifier is assumed to be steady and in cold conditions. The standard k-ε two-equation turbulence model was used with isotropic assumptions of the fluid flow. Therefore, general form of governing equation for this problem is as following:

$$\frac{1}{r}\frac{\partial}{\partial r}\left[r\left(\rho\phi V_r-\Gamma_\phi\frac{\partial\phi}{\partial r}\right)\right]+\frac{\partial}{\partial z}\left(\rho\phi V_z-\Gamma_\phi\frac{\partial\phi}{\partial z}\right)=S_\phi \qquad (1)$$

The meaning of ϕ, Γ_ϕ and source item S_ϕ of each conservation equations are shown in Table 1:

Table 1. Governing equations

Equation	ϕ	Γ_ϕ	S_ϕ
Continuous	1	0	0
Axial momentum	V_r	μ_{eff}	$-\frac{\partial p^*}{\partial r}-\frac{2\mu_{eff}V_r}{r^2}+\frac{1}{r}\frac{\partial}{\partial r}\left(r\mu_{eff}\frac{\partial v_r}{\partial r}\right)+\frac{\partial}{\partial z}\left(\mu_{eff}\frac{\partial V_z}{\partial r}\right)$
Radial momentum	V_z	μ_{eff}	$-\frac{\partial p^*}{\partial z}+\frac{1}{r}\frac{\partial}{\partial r}\left(r\mu_{eff}\frac{\partial V_r}{\partial z}\right)+\frac{\partial}{\partial z}\left(\mu_{eff}\frac{\partial V_z}{\partial z}\right)$
Turbulent kinetic energy	k	$\frac{\mu_{eff}}{\sigma_k}$	$G_k-C_D\rho\varepsilon$
Kinematics rate of dissipation	ε	$\frac{\mu_{eff}}{\sigma_\varepsilon}$	$\frac{\varepsilon}{k}\left(C_1G_k-C_2\rho\varepsilon\right)$

Where: $G_k=\mu_{eff}\left\{2\left[\left(\frac{\partial V_r}{\partial r}\right)^2+\left(\frac{\partial V_z}{\partial z}\right)^2+\left(\frac{V_r}{r}\right)^2\right]+\left(\frac{\partial V_r}{\partial z}+\frac{\partial V_z}{\partial r}\right)^2\right\}$

$$\mu_{eff}=\mu+\mu_t \qquad \mu_t=c_\mu\rho\frac{k^2}{\varepsilon} \qquad (2)$$

Table 2. Some constants used in the program

C_1	C_2	C_D	C_μ	σ_k	σ_ε
1.44	1.92	1.0	0.09	1.0	1.3

3.2 Boundary conditions

3.2.1 Inlet: The Oxygen is blowing in from the off-axis position (shift, not from center hole of the burner) and the high inlet velocity U_{in} is assumed to be uniform distribution. Meanwhile, the velocity components of axial (V_z) and radial (V_r) are also considered about the inject angle α. The inlet turbulence kinetic energy k and dissipation rate ε are set as follows:

$k_{in}=\frac{1}{2}I_{in}^2U_{in}^2$, where I_{in} means the velocity ratio of fluctuant to mean, here set $I_{in}=10\%$.

$\varepsilon_{in}=\frac{K_{in}^{1.5}}{(0.005R)}$, where R means the radius of the computation field.

3.2.2 Outlet: The flow field is assumed to be fully developed at the outlet of the gasifier. In order to meet the needs for mass conservation, the velocity at the outlet was corrected by the flow rate of the gas.

$$\left(\frac{\partial \phi}{\partial z}\right) = 0\,,\ \phi = V_z\ V_r\ P,\ k,\ \varepsilon \tag{3}$$

3.3.3 Axis: the characteristic of the flow field is symmetrical, which means $\left(\frac{\partial \phi}{\partial r}\right) = 0,\ \phi = V_z V_r P, k, \varepsilon$.

3.2.4 Wall: The wall function was used to deal with the points near the wall. By defining the local Reynolds number:

$$Y^{*} = \rho C_{\mu}^{1/4} K_{P}^{1/2} y_{P} / \mu \tag{4}$$

The shear stresses near the wall can be expressed as:

$$\tau_{w} = \mu \frac{V_P}{y_P} \qquad Y^{+} \le 11.63$$

$$\tau_{w} = \frac{\beta \rho C_{\mu}^{1/4} K_{P}^{1/2} V_P}{\ln(E Y^{*})} \qquad Y^{+} > 11.63 \tag{5}$$

where y_p is the distance of the point P to the wall, β=0.4187, E=9.793.

3.3 Solving method

During simulation work, the flow field was divided into non-uniform grids. The governing differential Equation (2) was integrated over each control volume. The momentum equations were specially treated by using the staggered grid system, while the pressure-correction equation was applied to avoid false diffusion. The power-law scheme is used for convection-diffusion formulation and the line-by-line TDMA method was used for the discrete algebraic equations. The SIMPLE algorithm suggested by S.V.Patanka (1980)[2] was applied for the whole iteration procedure.

4. Calculation results

During the calculation, the computer model was built from simple to complex gradually, which means consider many of the various influence parameters one by one. In order to discuss the characteristic of the flow field at different situations, the flow field inside the 0.5T/D old gasifier was simulated at first. The calculation region of the gasifier in KIER was assumed to be $R=0.1m$ with a length $L=2.05m$. The region was divided into 51×30 non-uniform grids with extremely fine grids at the burner inlet region.

As to the designed operate conditions, the O_2 is injected in from the shift (off-axis) position with an inlet angle of $\alpha=-15°$ at a high input velocity U_{in}. The calculation results of the flow field are shown as Fig.3~Fig.8.

4.1 The velocity field inside the gasifier

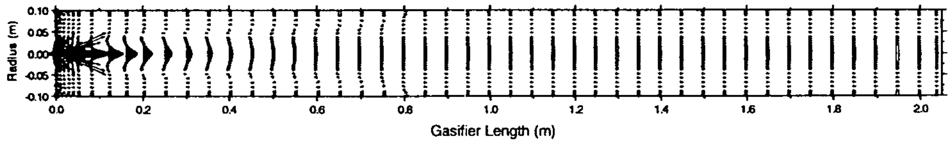


Fig.3 The velocity vectors inside the entrained flow gasifier

Fig.3 is the simulation result of velocity distribution inside the reactor. From here we can see, when the gas injects in, it will expand and form a parabolic velocity profile with maximum magnitude near the centerline. Because of the very fine grids at inlet region, in order to see the velocity at this small area more clearly, an enlargement of the Fig.3 are shown in Fig.4. From it one can see that actually when the oxygen blowing in with an angle, the maximum velocity is not at the center at first while after a short distance, the velocity becomes a parabolic shape profile. Meanwhile, there is a small internal recirculation zone at the burner inlet region. When the gasifier length great than 1.0m, the flow field becomes uniform with small change along the axis and the influence of the gas blowing seems not significant. An external recirculation zone is formed with its influence length less than 1.0m along the gasifier. Fig.5 is an illustration of streamline distribution of the flow field. From it we can see the external recirculation more clearly. Also one can see that the whole flow field appears like a pair of scissors when blowing in at high speed and this is reasonable with design purpose, but the expansion of flow is not very large due to the very small inlet position.

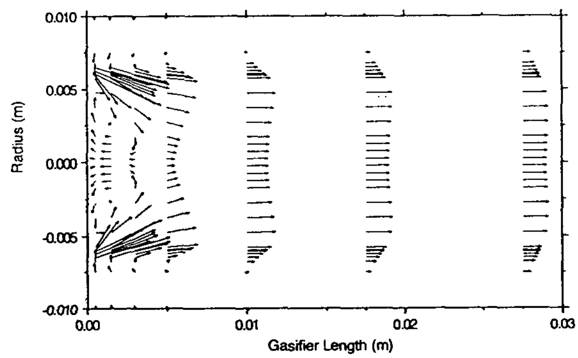


Fig. 4 Enlargement of the burner inlet region

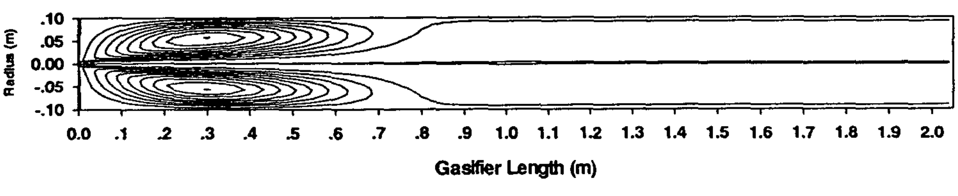


Fig.5 The stream line of the flow inside the gasifier

4.2 The Pressure, *k* and ε distribution inside the gasifier

The simulation results also give out the detailed information of pressure P, turbulent kinetic energy k and the kinematics rate of dissipation ε distribution in the flow field. From the contour graph of pressure field in Fig.6 we can see, the pressure changes acutely at the small region of burner inlet and become nearly uniform after the length of gasifier exceeds 0.6m. The turbulent kinetic energy *k* reflects the turbulence magnitude of the flow, it can be seen from Fig.7 that the large value of *k* are mainly concentrated near the gas inlet region, which means when the oxygen blows in, the turbulence is violent at this region. The similar trend can also be found in Fig.8 of ε distribution. The kinematics rate of dissipation ε profile indicates that the main dissipation of the inlet gas energy is also taken place near the small inlet region.

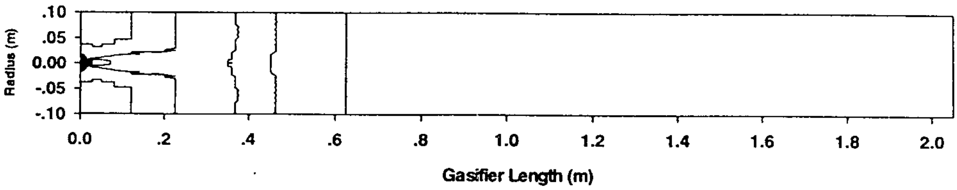


Fig.6 The pressure field distribution inside the gasifier

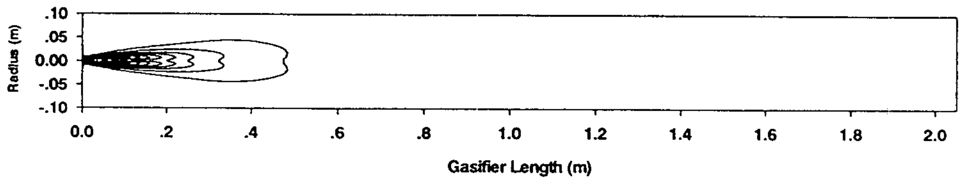


Fig.7 The turbulent kinetic energy k distribution contour inside the gasifier

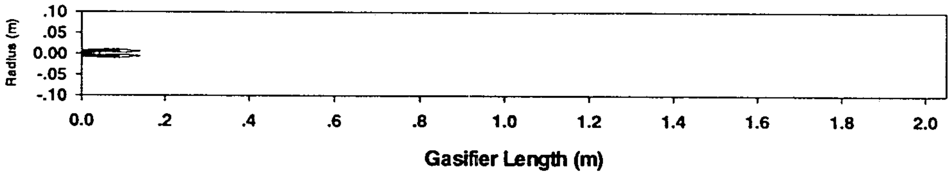
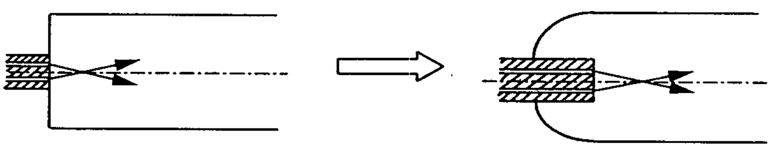


Fig.8 The kinematics rate of dissipation ε distribution contour inside the gasifier

4.3 Influence of the burner extending in length and round boundary

Till now in order to simply the problem, we did the calculation based on the inlet part as shown in Fig.9 (a). But actually the inlet part geometry scheme are as Fig.9 (b) in real designing. Therefore, we modified the boundary conditions and to see what is the difference between these two cases.



(a) (b)

Fig.9 Inlet scheme of the KIER gasifier

In simulation work, the grid system was redefined and the round edge boundary was treated as step shape. The calculation result is shown in Fig.10. From the results we can see, the basic shape of the flow field is nearly the same as that in Fig.3. The burner extending in length and the round edge do not influence the flow field so much. This gives us some information that sometimes roughly assumptions or simplifications can also give us some useful information about the engineering projects. Fig. 10 can serve as the final result of the flow field inside the old coal gasifier.

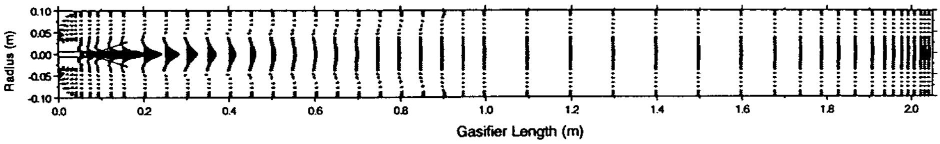


Fig.10 Influence of the inlet geometry to the flow field

4.4 Influence of gasifier geometry parameters

The designed geometry parameters of 1.0T/D new gasifier in KIER are different with the 0.5T/D old gasifier. The main difference is that the length of the gasifier decreases to 1.0m while a little enlargement in the inner diameter. The calculation results for this case are shown in Fig.11.

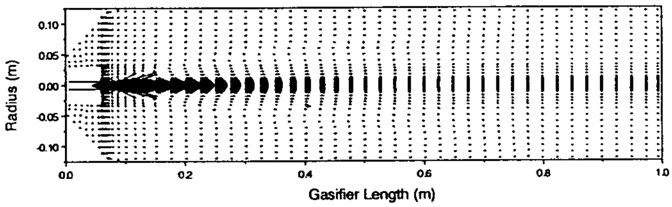


Fig.11 Cold flow field inside 1T/D entrained flow gasifier

Comparing Fig.11 with Fig.10 we can see, the flow filed of the 1.0T/D new gasifier is much similar as the 0.5T/D gasifier in the length of 0~1.0m.

5. Conclusions and Discussions

In order to know more detail about flow field inside KIER gasifier, a serial of comparisons between different parameters and operating conditions have been done during computation. From the simulation results we can draw the following conclusions:

(1) The basic patterns of the flow field inside the gasifier at different conditions are nearly the same with a parabolic distribution. When oxygen blows in, it expands with the larger velocity at the center region and smaller velocity near the wall. There exists an obvious external recirculation region which was caused by the injecting from center. The length of the recirculation zone is less than 1.0m both at the old gasifier and new gasifier conditions. After the gasifier length great than 1.0m, the flow becomes uniform.

(2) The different inlet velocity and gas injecting in angles don't influence the basic flow pattern inside the gasifier too much, while the geometry parameters of the burner, such as the Oxygen injecting in position and the inlet diameter will influence the flow field greatly. However, the value of these parameters are very small comparing with the space inside gasifier (for example, the ratio of gas inlet diameter to the gasifier diameter is about $d/D=1.28/200=0.64\%$). As a result, even we changed some parameters in calculation, the influence regions are so small that the difference between whole flow field can be neglected.

From the above conclusions from calculation, further discussions may be made. If the Reynolds number of fluid dynamics $Re=\rho UD/\mu=UD/\nu$ is introduced, as we know, the dimensionless parameter Re is the governing parameter for fluid viscous flow. From here we can see that the major parameters which will influence the flow field is velocity U, geometry parameter D and the fluid viscosity ν. In our situation, the gasifier diameter is about $D=0.1m$, and the fluid is Oxygen gas with a viscosity $\nu=1.8\times10^{-5}m^2/s$ at cold conditions. If the gas inlet velocity $U_{in}=1m/s$, the Re will be $Re\sim5500$. This means the flow has entered the region of fully turbulence. From the fluid mechanics knowledge we know that at this region ($Re\sim2300\sim1.0\times10^7$), the friction coefficient of the flow nearly keeps the same value. This means that even if the Re increase, the characteristic of the flow will not change. As to the flow inside KIER gasifier, even if the velocity increase from $U_{in}=1m/s$ to $U_{in}=255m/s$, the Re will increase from $Re\sim5500$ to $Re\sim1.5\times10^6$, the flow is still in the fully turbulence region. This analysis can explain some of our conclusions during the simulation.

In KIER gasifier, the fluid properties and inlet flow rate are given (this means ν and U are determined), the only thing will change the Re is the burner geometry parameters. But as discussed above, the burner geometry parameters are very small compare with the space inside the gasifier. Even if we change these parameters a little (not to an extent that will cause qualitative change), the influence on the Reynolds number is not significantly. As a result, our calculations of changing burners parameters such as inlet diameter, inlet angle and inlet position seem do not change the flow field too much.

The purpose of calculation on the flow field is to help people know more clearly of gasification process inside the gasifier, especially to know how the flow pattern will influence the coal slurry particles trajectories, the behavior of the coal slurry particles as well as its distribution, residence time, etc. Through the parameter analysis, it can help us to optimum the burner design in order to discrete the coal slurry uniformly and form a well-mixed particle distribution. Based on this work, the coal gasification model with two-phase turbulent flow, combustion and gasification, heat transfer will be built for KIER 1.0T/D entrained-flow new gasifier.

Acknowledgements

The authors would like to thank KOSEF for financial support of this work under Korea-China Young Scientist Exchange Program Training Fellowship.

References

1. D.B.Spalding. Numerical prediction of flow, heat transfer, turbulence and combustion. New York: Pergamon Press Ltd., 1983.
2. S.V.Patankar. Numerical heat transfer and fluid flow. McGraw-Hill Book Co., 1980.
3. C.Arakawa. Computational fluid dynamics for engineering. University of Tokyo Press, 1994.
4. X. J. Liu, T. J. Park, Numerical simulation of the flow field in an entrained flow gasifier, 2nd China-Korea Joint Workshop on Coal and Clean Energy Utilization Technology, Nov. 2-6, 1998, Taejon.
5. X.J.Liu. Numerical simulation of coal gasification. Post-doc report, KIER, 1999.
6. T.J.Park, J.H.Kim, et al. Development of entrained-flow coal gasifier, The Latin-American congress: Electricity generation and transmission, Nov. 9-13, 1999, Brazil.
7. T.J.Park, J.H.Kim, et al. Status of coal gasification R&D in KIER, 1st China-Korea joint workshop in coal utilization technology, Sep. 2-6, 1996, Beijing.