Korean J. Chem. Eng., **18**(3), 376-381 (2001)

Numerical Analysis of the Flow Field inside an Entrained-Flow Gasifier

Young Chan Choi†, Xiang Yang Li*, Tae Jun Park, Jae Ho Kim and Jae Goo Lee

Energy & Environment Research Division, Korea Institute of Energy Research (KIER),
P.O.Box 103, Yusong-gu, Daejeon 305-343, Korea
*Thermal Engineering Department, Tsinghua University, Beijing 100084, China
(*Received 15 January 2001 • accepted 13 April 2001*)

Abstract−The flow field of an entrained-flow gasifier was numerically simulated to describe coal gasification process. The standard k-ε turbulence model and SIMPLE procedure were used with the Primitive-Variable method during computation. In order to investigate the influencing factors on the flow field that may have a great effect on coal gasification process, some parametric studies were performed by changing the gas injection angle, gas inlet diameter, gas inlet velocity, extension in burner length and gasifier geometry. The calculation results showed that the basic patterns of the flow field inside the gasifier were nearly the same with a parabolic distribution irrespective of the change in parameters. There existed an obvious external recirculation zone with axial length less than 1.0 m and a narrow internal recirculation region was observed in the entrance of gasifier inlet. The geometry parameters of the burner, such as the oxygen inlet diameter and angle, influenced the flow field at the inlet region near the burner. But after a certain length along the gasifier, the flow field was nearly the same as that in the basic case.

Key words: Numerical Simulation, k-ε Turbulence Model, Flow Field, Entrained-Flow

INTRODUCTION

Coal gasification is a key process for Integrated-coal Gasification Combined Cycles (IGCC) power plant. Being one of the most competitive and promising coal gasification technologies, the slurry feed type entrained-flow coal gasifier has been extensively studied in the Korea Institute of Energy Research (KIER) and some important achievements have been gained by investigating the complicated nature of coal characteristics and establishing the optimal operating condition using the 1 T/D bench-scale, entrained flow coal gasifier [Park, 1999].

In an entrained flow slagging coal gasifier using coal water slurry, it should be necessary to maintain the uniform mixing among the coal-water slurry with oxygen to obtain the higher carbon conversion in short residence time (0.4-5 s). A calculation study on the flow field inside the gasifier is to give a better understanding of the gasification process, especially to know how the flow pattern will influence the coal slurry particles trajectories and the behavior of the coal slurry particles as well as its distribution, residence time, etc. Furthermore, the burner design through the parametric studies can be optimized in order to discrete the coal slurry uniformly and form a well-mixed particle distribution.

With the rapid development of computer technology in recent years, numerical simulation of some complex flow fields has been possible. At present, some general codes or commercial software are available to simulate the common flow field. But due to the complicated nature of turbulence and different initial and boundary conditions, it still needs special considerations and modifications for each engineering object if careful and detailed research is needed. Numerous simulation works on the coal gasification in the different type of reactor have been implemented in order to predict the coal gasification performance [Arakawa, 1994; Govin, 1984; Lee, 2000; Pantankar, 1980; Spalding, 1983; Yoon, 1985]. However, a few simulation works to analyse the flow field have been published. A previous simulation work at KIER calculated the flow field of an entrained flow gasifier by using the Vorticity-Stream Function method [Liu, 1999]. Assuming the inlet coal slurry to be some kind of primary air with the same momentum, the calculation result showed that owing to the high injection velocity of the secondary flow of oxygen, there was an obvious central recirculation zone near the burner.

In this paper, the flow field inside the entrained flow gasifier in KIER was predicted by using the Primitive Variable Approach in order to meet the needs for optimum design of a new reactor and burner. Some parametric studies were performed by changing the gas injection angle, gas inlet diameter, gas inlet velocity, extension in burner length and gasifier geometry. Therefore, the calculation of the flow field inside the gasifier at different design conditions can help to determine if the coal slurry particles can mix well or distribute better inside the gasifier and optimum design of a new reactor and burner.

PHYSICAL MODEL

The schematic diagram of the experimental oxygen-blown, entrained flow coal gasifier and the detailed structure of the burner are presented in Figs. 1 and 2, respectively. As shown in Fig. 2, the coal-water-slurry (CWS) is injected into the gasifier through the center hole of the burner, while the oxygen is blown in through eight surrounding holes during gasification. The burner is designed so that there is an injection angle (α: 15°) for the secondary oxygen flow. Therefore, as soon as the slurry is fed into the gasifier, it is impacted by the high speed secondary oxygen flow and divided

†To whom correspondence should be addressed.
E-mail: youngchan@kier.re.kr

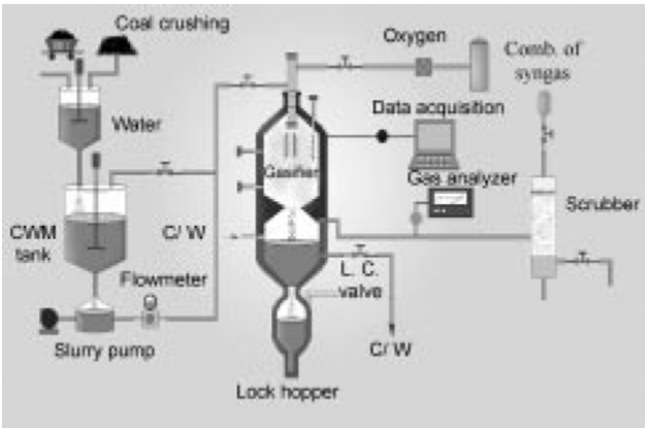


Fig. 1. PFD of the oxygen-blown, entrained flow coal gasifier.

into droplets of different sizes. The main geometric parameters of the gasifier and burner are listed in Table 1.

For simplicity, 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 the simulation. Therefore, this three-dimensional geometry can be reduced to a two-dimensional axial-symmetric case. The influence of coal slurry injection was not considered in this flow field simulation, therefore the center hole was assumed to be solid and the surface was treated as a solid wall. But in the real coal gasification process, the coal slurry was injected in through the center hole and the solid phase will greatly influence the gas flow field and this effect must be taken into account.

MODEL DESCRIPTION

1. Governing Equations

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

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

The meanings of ϕ, Γ_ϕ and source item S_ϕ of each conservation equations are shown in Table 2.

2. Boundary Conditions

2-1. Inlet

The oxygen is blowing in from the off-axis position (not from center hole of the burner) and the high inlet velocity U_{in} and the inlet

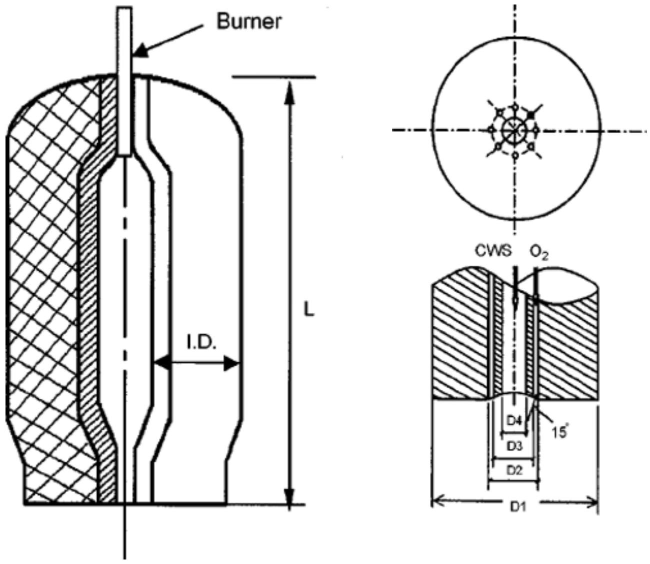


Fig. 2. Schematic of experimental gasifier and detailed structure of the burner.

Table 1. Some geometry parameters of gasifier and burner (mm)

1 T/D gasifier		Burner			
I.D.	Length	D1	D2	D3	D4
200	2,050	60	13.28	12	3

Table 2. ϕ, Γ_ϕ and source item S_ϕ of each conservation 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	$\frac{\mu_{eff}}{\sigma_\varepsilon}$	$\frac{\varepsilon}{k}(C_1G_k-C_2\rho\varepsilon)$

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

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

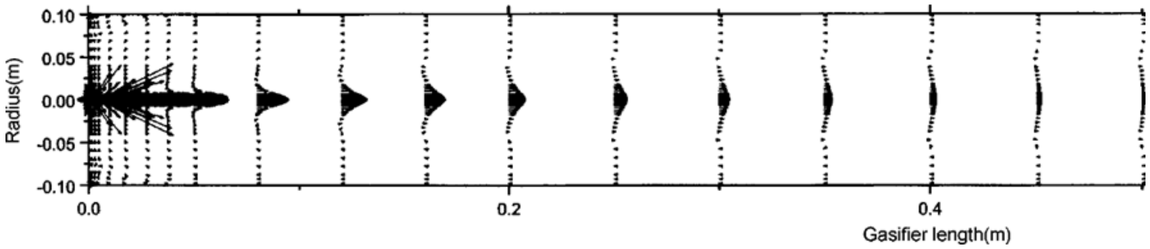


Fig. 3. Simulation result of velocity distribution inside the gasifier.

pressure P_{in} are assumed to be uniform distribution. Meanwhile, the axial (V_z) and radial (V_r) velocity components are also considered about the injection 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 fluctuation to mean.

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

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.

$$\left(\frac{\partial\phi}{\partial z}\right)=0,\ \phi=V_z,\ V_r,\ P,\ k \text{ and } \varepsilon \quad (2)$$

2-3. Axis

The characteristic of the flow field is symmetrical, which means

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

2-4. Wall

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

$$Y^+=\rho C_\mu^{1/4}K_p^{1/2}y_p/\mu \quad (3)$$

The shear stresses near the wall can be expressed as:

$$\tau_w=\mu\frac{V_p}{y_p} \qquad Y^+\le 11.63$$

$$\tau_w=\frac{\beta\rho C_\mu^{1/4}K_p^{1/2}V_p}{\ln(EY^+)} \qquad Y^+>11.63 \quad (4)$$

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

3. Solution Method

During simulation work, the flow field was divided into non-uniform grids. The governing differential Equation in Eq. (1) 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 Patanka [1980] was applied for the whole iteration procedure.

Table 3. Some constants used in the program

C_1	C_2	C_D	C_μ	σ_k	σ_ε	U_{in}	P_{in}
1.44	1.92	1.0	0.09	1.0	1.3	255 m/s	1 atm

RESULTS AND DISCUSSION

During the calculation, the computer model was gradually made more complex, which meant considering many of the various influence parameters one by one. The region was divided into 51×30 non-uniform extremely fine grids at the burner inlet region. As to the designed operating conditions, the O_2 is injected in from the off-line position with an injection angle of $\alpha=15^{\circ}$ at a high input velocity U_{in}. Some other constants used in the model are shown in Table 3.

1. Velocity Field Inside the 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 the inlet region in order to see the velocity distribution in this small area more clearly, an enlargement of Fig. 3 is shown in Fig. 4. As shown in Fig. 4, when the oxygen blows in at an angle, the maximum velocity is not at the center at first while after a short distance, the velocity assumes a parabolic shape profile. Meanwhile, there is a small internal recirculation zone at the burner inlet region. When the gasifier length is greater than 1.0 m, the flow field becomes uniform with little change along the axis and the influence of the gas blowing appears not to be significant. An external recirculation zone is formed

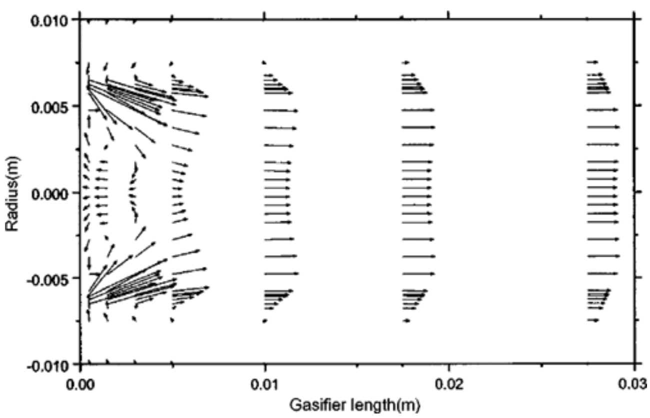


Fig. 4. Enlargement of the burner inlet region in Fig. 3.

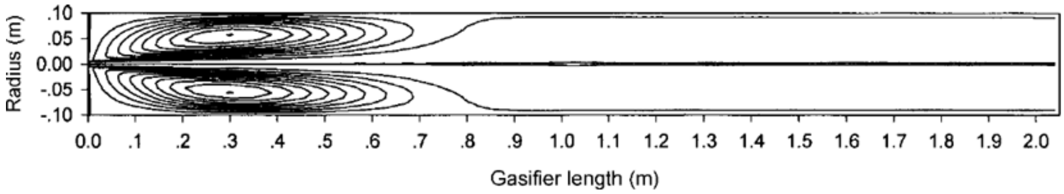


Fig. 5. Stream line of the flow inside the gasifier.

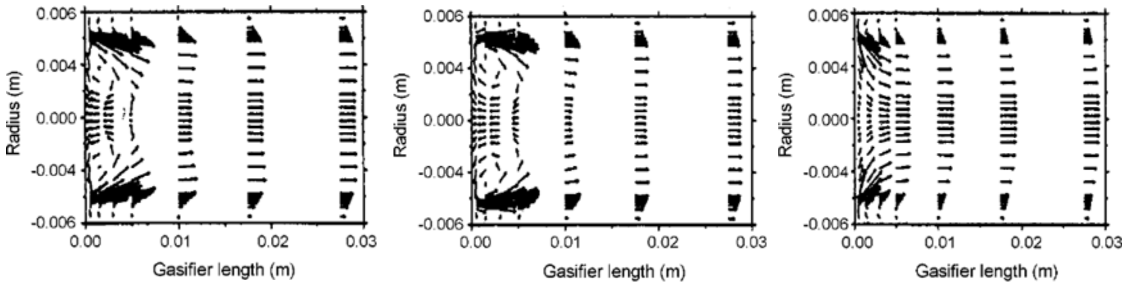
along the gasifier with its influence length less than 1.0 m. Fig. 5 is an illustration of the streamline distribution of the flow field. It shows the external recirculation more clearly and the whole flow field appears like a pair of scissors when blowing in at high speed. This is reasonable for design purposes, but the expansion of flow is not very large due to the very small inlet position.

2. Influence of the O_2 Injection Angle

The designed O_2 injection angle on the burner is $\alpha=15°$ in KIER gasifier. In order to examine the influence of inlet angle on the flow field, the O_2 injection angle is changed from $\alpha=0°$, $\alpha=-15°$ to $\alpha=45°$ respectively. The calculation results are shown in Fig. 6. It can be seen from Fig. 6 that there are some differences at the inlet region near the burner and centerline zone when the gas blows in from different angles. But as to the whole flow, the flow field does not change much with different O_2 injection angle.

3. Influence of the O_2 Injection Diameter

According to the design data, the O_2 injection diameter is only 1.28 mm (calculated from the position 13.28 mm-12mm). We enlarged its diameter from 1.28 mm to 4 mm (which means the oxygen injection positions are from D=12 mm to D=16 mm) with the other conditions remaining unchanged. The calculation results are shown in Fig. 7. It shows that there is a big difference at the inlet



(a) $\alpha = 0°$ (b) $\alpha = -15°$ (c) $\alpha = 45°$

Fig. 6. Influence of the O_2 injection angle on the flow field.

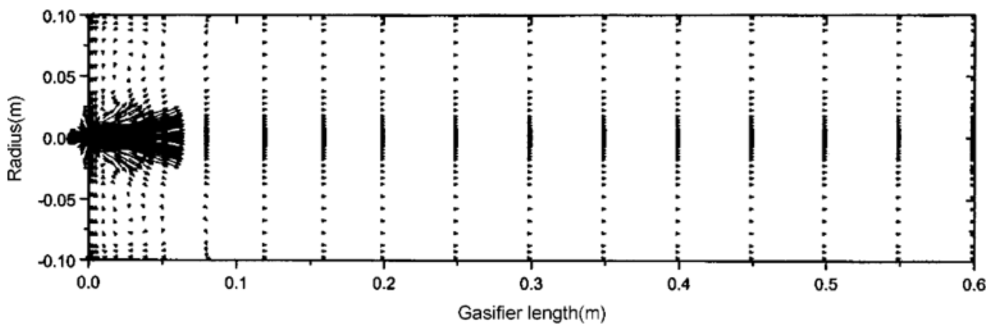


Fig. 7. Influence of the O_2 inlet diameter enlargement on the flow field.

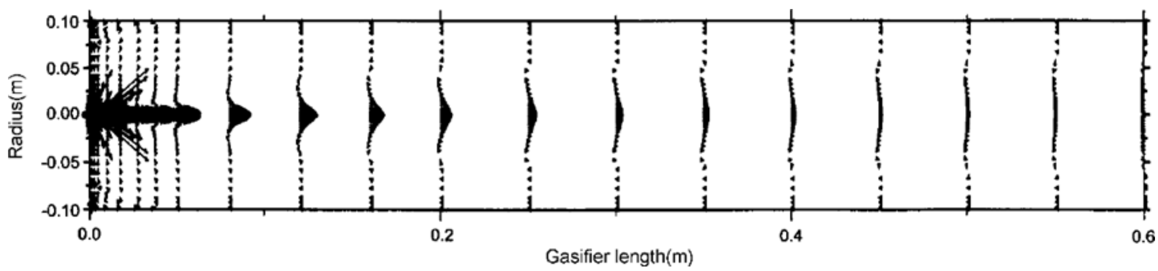


Fig. 8. Influence of the O_2 injection velocity on the flow field.

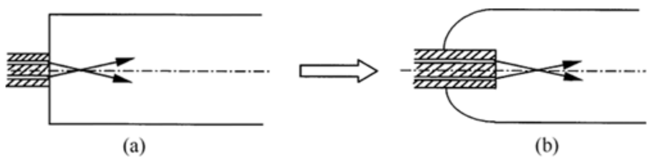


Fig. 9. Schematic diagram of burner position.

region comparing with Fig. 3. The gas inlet diameter will change the flow field especially at the inlet region near the burner. However, still we can see that after a certain length along the gasifier, the flow field is nearly the same as that in Fig. 3.

4. Influence of the Inlet Velocity

The above calculations are based on the designing inlet velocity of U_{in}=255 m/s. In order to investigate the influence of the inlet velocity on the flow field, a wide range of inlet velocities which are from 1 m/s to 273.25 m/s were examined during simulation parameter study. Fig. 8. is the case when the inlet velocity decreases to 1 m/s. The calculation result shows that the basic shape of flow field are nearly the same as that in Fig. 3, except that the magnitude of velocity components, pressures and other parameters of the flow field will certainly decrease correspondingly at each grid points.

5. Influence of the Extension in Burner Length and Round Boundary

In order to simplify the problem, we did the calculation based on the inlet part as shown in Fig. 9(a). But actually in the real design, the geometric schematic of the inlet part is as shown in Fig. 9(b). Therefore, we modified the boundary conditions to see the difference between these two cases. In the 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 that the basic shape of the flow field is nearly the same as that in Fig. 3. The extension in burner length and round edge boundary do not influence the flow field greatly. Fig. 10 can serve as the final result of the flow field inside the gasifier.

6. Influence of Gasifier Length

The designed length of the entrained flow gasifier in KIER is 2.05 m. As obtained results from the experiments, every gasification process are finished within a length less than 0.5 m. In order to see the effect of gasifier length decrease on the flow field, we did the calculation for the case of gasifier length of 1 m. The calculation results for this case are shown in Fig. 11. Comparing Fig. 11 with Fig. 10 we can see that the flow filed did not change greatly.

CONCLUSIONS

To investigate more detail about the flow field inside an entrained flow gasifier, some parametric studies were performed by changing the gas injection angle, gas inlet diameter, gas inlet velocity, extension in burner length and gasifier geometry. 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 injection from the center. The length of the recirculation zone is less than 1.0 m and the flow becomes uniform with a gasifier length greater than 1.0 m.

2. The results show also that the basic flow pattern inside the gasifier do not change greatly when the parameters are changed. The geometric parameters of the burner, such as the oxygen inlet diameter and injection angle, influenced the flow field greatly at the inlet region near the burner. But after a certain length along the gasifier, the flow field is nearly the same as that in the basic case.

REFERENCES

Arakawa, C., "Computational Fluid Dynamics for Engineering," Uni-

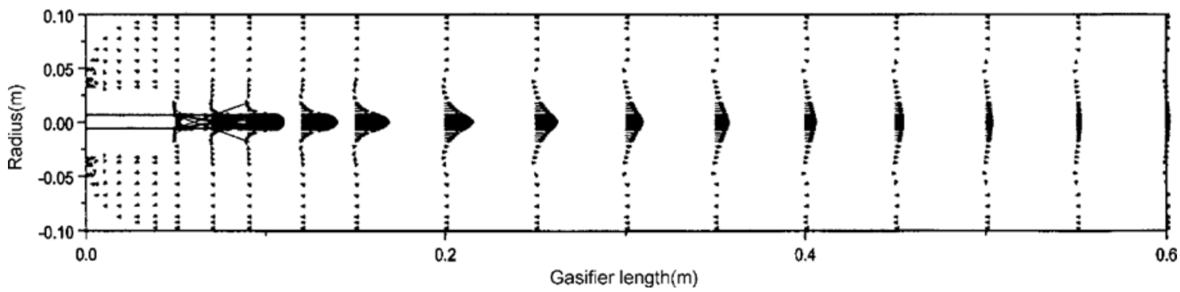


Fig. 10. Influence of the burner inlet geometry on the flow field.

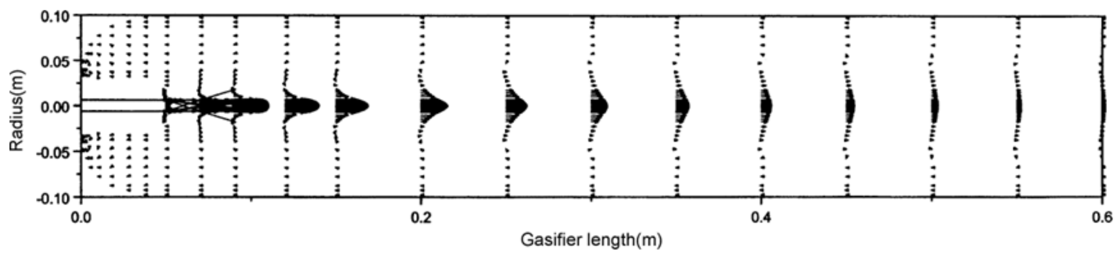


Fig. 11. Influence of gasifier geometry on the flow field.

versity of Tokyo Press (1994).
Chang, Y. H., "A Mathematical Modeling for the Gasification of Single Coal Particle," *Korean J. Chem. Eng.*, **23**, 273 (1985).
Govin, R. and Shah, J., "Modeling and Simulation of an Entrained Flow Coal Gasifier," *AIChE Journal*, **30**, 79 (1984).
Lee, J. M., "Modeling and Coal Gasification in an Internally Circulating Fluidized Bed Reactor," *HWAHAK KONGHAK*, **38**, 259 (2000).
Liu, X. J., "Numerical Simulation of Coal Gasification," Post-doc report, KIER (1999).
Park, T. J., Kim, J. H., Lee, H. J. and Lee, J. G., "Development of Entrained-Flow Coal Gasifier," The Latin-American Congress: Electricity Generation and Transmission, Nov. 9-13, Brazil (1999).
Patankar, S. V., "Numerical Heat Transfer and Fluid Flow," McGraw-Hill Book Co. (1980).
Spalding, D. B., "Numerical Prediction of Flow, Heat Transfer, Turbulence and Combustion," New York, Pergamon Press Ltd. (1983).