Numerical Investigation on Influence of Fan Speed and Swirling Gas Injection on Thermal-Flow Characteristics in Nitrocarburizing Furnace

Hyung Ju Lee, Jaeyong Park and Seong Hyuk Lee

School of Mechanical Engineering, Chung-Ang University, Seoul 156–756, Korea

This study was carried out to analyze the flow and heat transfer characteristics in a furnace, and to evaluate the influence of the two major design factors for improving the flow mixing and heat transfer. The grid system was constructed using ANSYS ICEM (V.17.0), and numerical simulations were conducted using the commercial CFD code (ANSYS Fluent V.17.0). The fan plays an important role in flow mixing and convective heat transfer in the furnace. Accordingly, temperature uniformity was improved with an increase in the fan speed owing to an enhancement of the convective heat transfer. Moreover, when swirl flow was applied to the inlet gas, temperature uniformity was improved in the lower section of the furnace. This is because the swirl flow of the inlet gas improved the flow mixing in the vicinity of the gas inlet where it is not affected by the rotational force of the fan. Controlling the swirl intensity of the inlet gas with lower fan speeds is more energy efficient because flow mixing and temperature uniformity could be improved sufficiently.

(Received May 9, 2017; Accepted July 3, 2017; Published August 4, 2017)

Keywords: CFD (computational fluid dynamics), fan speed, heat treatment furnace, nitrocarburizing, swirl number, temperature uniformity

1. Introduction

Machine components used in various applications, such as machinery, automobiles, and power plants, are exposed to extreme temperatures and pressures, which directly affect their reliability and sustainability. Also, the demand for surface-treatment techniques is gradually increasing to resolve these problems. Among these techniques, the nitriding and nitrocarburizing processes, which introduce nitrogen atoms or both nitrogen and carbon atoms, respectively, into the surface of a component, have been the most effective for many decades. A valuable consequence of these techniques is the reduced distortion and deformation of the heat-treated parts because they are lower-temperature methods compared to other heat treatment methods. Meanwhile, these processes also improve wear resistance, corrosion resistance, and fatigue endurance. In particular, these resistance performances are closely related to the nitriding layer thickness determined by the operating factors, such as atmosphere’s gas composition, temperature, and process time.

Much research has been conducted investigating the nitriding as well as the nitrocarburizing methods. Hassani-Gangaraj and Guaglio analyzed the reaction-diffusion mechanism for nitrogen and quantified the Fe-N phase thermodynamically. The growth rate of a compound layer was also predicted and examined with the use of the commercial finite element code (ABAQUS). Du et al. analyzed the material characteristics in relation to the thickness and composition of the compound layer, as well as various important parameters such as nitriding potential and carburizing potential. The process time was considered to affect the quality of nitrocarburizing. Horak et al. proposed a mathematical model for predicting the microhardness of the nitrided steel, and compared this model with the experimental results in relation to the atmosphere gas temperatures and pressures. The hardness of the nitriding layer was measured with respect to time and temperature, and it was found that a multi-nitriding method could improve the hardness of the nitriding layer. In addition, a number of factors such as nitriding potential, carburizing potential, process temperature, and process time were analyzed to enhance the quality of the heat treatment under different operating conditions for the nitriding and nitrocarburizing processes. In fact, a number of studies have been conducted to quantitatively analyze the various parameters such as layer thickness, hardness, porosity, material properties, amongst others, in evaluating the quality of heat treatment. These parameters are strongly affected by atmosphere gas temperature and composition in the furnace. The nitriding thickness of the nitriding layer specifically may change with a change of location in the furnace. This is one of the serious flaws in the nitrocarburizing process, as the non-uniform thickness of the nitriding layer, which could lead to deterioration of quality, is affected by local flow conditions as well as the temperature and composition of the atmosphere gas. There has, however, been limited research on how flow and heat characteristics in the furnace affect material characteristics. These problems need to be resolved to allow efficient designs. However, there are limits to experimentation in a furnace with a very high gas phase temperature and toxic gases. In particular, it would be difficult to directly measure flow velocity and gas temperature fields inside the furnace during actual operation. Therefore, with computational fluid dynamics (CFD) being one of the design methods, this study aims to numerically investigate the flow and heat transfer characteristics in the industrial furnace. In particular, the temperature distribution in the furnace will be analyzed with respect to two important design factors, namely fan speed and swirling gas injection on flow mixing.

2. Numerical Simulation

2.1 Governing equations

To analyze the thermal flow characteristics of the atmosphere gas, the continuity, Reynolds-averaged Navier-Stokes, Reynolds stress transport, energy, and species transport equations were calculated as shown in eq. (1)–(5) respectively.
\[ \frac{\partial}{\partial x_i}(\rho u_i) = 0 \]  

\[ \frac{\partial}{\partial x_k} (\rho u_i u_j) = -\frac{\partial p}{\partial x_k} + \frac{\partial}{\partial x_j} \left[ \rho \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_l}{\partial x_l} \right) \right] + \frac{\partial}{\partial x_k} \left( -\rho u_i u' u'_k \right) + S, \]  

\[ \frac{\partial}{\partial x_k} \left( \rho u_i u'_k \right) = D_{ij} + P_{ij} + \Pi_{ij} + \Omega_{ij} - \varepsilon_{ij}, \]  

\[ \frac{\partial}{\partial x_k} [u_i (\rho E + p)] = \frac{\partial}{\partial x_j} \left[ \left( k + C_p \mu_i \right) \frac{\partial T}{\partial x_j} - \sum \delta_{ij} J_i \right] + S_h, \]  

\[ \nabla \cdot (\rho \Phi) = -\nabla \cdot J, \]  

\[ D_{ij} = -\frac{\partial}{\partial x_k} \left[ \rho u_i u'_k + p' (\delta_{ik} u'_j + \delta_{jk} u'_i) \right], \]  

\[ P_{ij} = -\rho \left( \frac{u_i u'_k}{\delta_{ik}} \frac{\partial u_k}{\partial x_j} + \frac{u'_i u'_k}{\delta_{ik}} \frac{\partial u_k}{\partial x_j} \right), \]  

\[ \Pi_{ij} = \rho' \left( \frac{\partial u'_i}{\partial x_j} + \frac{\partial u'_j}{\partial x_i} \right), \]  

\[ \Omega_{ij} = -2 \rho \Omega_k \left( u'_i u'_m \Phi_{km} + u'_i \delta_{mk} \Phi_{km} \right), \]  

\[ \varepsilon_{ij} = 2 \mu \left( \frac{\partial u'_i}{\partial x_j} + \frac{\partial u'_j}{\partial x_i} \right), \]

where \(-\rho u'_i u'_k\) is the Reynolds stress, \(D_{ij}\) is the transport of Reynolds stress by diffusion, \(P_{ij}\) is the rate of production of Reynolds stress, \(\Pi_{ij}\) is the transport of Reynolds stress due to pressure-strain interactions, \(\Omega_{ij}\) is the transport of Reynolds stress due to rotation, and \(\varepsilon_{ij}\) is the rate of dissipation of Reynolds stress. Furthermore, the discrete ordinates radiation model was considered. The incident radiation and the source term are calculated by

\[ \nabla \cdot (I(\vec{r}, \vec{s}) \vec{s}) = -(a + \sigma_s)I(\vec{r}, \vec{s}) + an \frac{\sigma T^4}{\pi} \]  

\[ + \frac{\sigma}{4 \pi} \int_0^{4 \pi} I(\vec{r}, \vec{s}') \Phi(\vec{s} \cdot \vec{s}') d\Omega', \]  

\[ S_h = -\frac{\partial q_c}{\partial x_i} = \alpha_i \left( 4\pi I_{ps}(\vec{r}) - \int_{4\pi} I(\vec{r}, \vec{s}) d\Omega' \right). \]

The multiple reference frame (MRF) method is adopted to simulate the rotation of the fan. It calculates the stationary frame and rotating frame separately, and virtual forces in non-inertial coordinate relative to the rotating frame are added to the source term of the Navier-Stokes equation. The source term, \(S\), is calculated as follows:

\[ S = -\rho \left( \frac{\vec{2} \Omega \times \vec{a} + \vec{\Omega} \times \vec{a} \times \vec{r}}{2} \right). \]

respectively. The present study was conducted by assuming incompressible flow, Newtonian fluid, and steady state, with the use of the commercial CFD code, FLUENT V. 17.016).

### 2.2 Geometry and grid system

This study investigates a pit furnace, which is widely used in nitrocarburizing processes, as shown in Fig. 1. The geometry was constructed in a three-dimensional computational domain which comprised main components such as a chamber, guide chamber, electric heater, amongst others. Atmospheric gas was distributed inside the chamber, and a guide chamber for controlling the internal flow was mounted inside the chamber. Electric heaters encircled the chamber, and a centrifugal fan for the stirring of internal flow was located under the furnace cover. Three gas inlets for atmosphere gases (ammonia, nitrogen, and carbon dioxide) and a gas outlet were included in the geometry. The height of the furnace was approximately 2.7 m, the chamber was 1.2 m in diameter, and guide chamber was 1.17 m in diameter. Also, the present study constructed tetrahedral meshes by using the ICEM-CFD (V.17.0) as shown in Fig. 2. The total grid number was finally taken as approximately 2,800,000 after grid independent tests.

### 2.3 Numerical details

This study analyzes the flow and heat transfer characteristics in the nitrocarburizing furnace. The boundary conditions used for the numerical CFD simulation are summarized in Fig. 3, and those were determined according to experimental
data and operating conditions provided by the real industry. To set the mass flow rates of ammonia, nitrogen, and carbon dioxide, the values were converted to velocities as shown in Table 1, and were set to 1.437, 0.663, and 0.11 m/s, respectively. The inlet gas temperature was set to 300 K and assumed as a single phase, with no phase changes. The inlet temperature of the electric heater was set to 883 K which was provided in the real industry, and the side wall with the exception of the electric heater was set to the adiabatic condition due to the use of insulation. In order to take into consideration the thermal losses at the furnace cover, the conjugate heat transfer between the solid substrate and the fluid was calculated. Additionally, the furnace cover temperature on the upper face was set to 413 K, the side face of the furnace cover was set to the adiabatic condition, and the bottom face of the furnace cover was set as a coupled wall for conjugate heat transfer. The atmosphere gas was initially filled with nitrogen. The MRF method was used to simulate the rotation of the fan, and its speed was set at 1,000, 2,000, and 3,000 rpm. Five different swirl numbers for the inlet gas were used to estimate the influence of swirling gas injection: 0, 0.5, 1, 3, and 5. Table 2 summarizes the different cases for this simulation.

3. Results and Discussion

3.1 Fan speed effect on thermal characteristics

In Fig. 4, the mean flow fields of atmosphere gases in the furnace are illustrated. From the rotational force of the fan, internal flow with high velocity passed between the chamber and the guide chamber walls, and recirculating flow was generated in the furnace. A strong swirl flow was formed in the lower section of the furnace, and it moved upward to the upper section of the furnace. The fan thus plays an important role in enhancing the flow mixing and convective heat transfer in the furnace. Figure 5 represents the axial velocity distributions of atmosphere gases in the YZ-plane for the three different fan speeds. A high velocity was estimated in the lower section of the furnace, and a recirculation region was locally formed in the vicinity of the furnace owing to local pressure differences. Axial and tangential velocity profiles at line 1 (y = 0.4 m) and line 2 (y = 2.2 m) are shown in Fig. 6.

Table 1 Boundary conditions of inlet gas.

<table>
<thead>
<tr>
<th>Inlet gas</th>
<th>Conditions</th>
<th>Details</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ammonia (NH₃)</td>
<td>Velocity inlet</td>
<td>1.437 m/s</td>
</tr>
<tr>
<td></td>
<td>Temperature</td>
<td>300 K</td>
</tr>
<tr>
<td>Nitrogen (N₂)</td>
<td>Velocity inlet</td>
<td>0.663 m/s</td>
</tr>
<tr>
<td></td>
<td>Temperature</td>
<td>300 K</td>
</tr>
<tr>
<td>Carbon Dioxide (CO₂)</td>
<td>Velocity inlet</td>
<td>0.110 m/s</td>
</tr>
<tr>
<td></td>
<td>Temperature</td>
<td>300 K</td>
</tr>
</tbody>
</table>

Table 2 Simulation cases.

<table>
<thead>
<tr>
<th>Case no.</th>
<th>Fan speed</th>
<th>Swirl number of inlet gas</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1</td>
<td>1,000 rpm</td>
<td>0</td>
</tr>
<tr>
<td>Case 2</td>
<td>2,000 rpm</td>
<td>0</td>
</tr>
<tr>
<td>Case 3</td>
<td>3,000 rpm</td>
<td>0</td>
</tr>
<tr>
<td>Case 4</td>
<td>1,000 rpm</td>
<td>0.5</td>
</tr>
<tr>
<td>Case 5</td>
<td>1,000 rpm</td>
<td>1</td>
</tr>
<tr>
<td>Case 6</td>
<td>1,000 rpm</td>
<td>3</td>
</tr>
<tr>
<td>Case 7</td>
<td>1,000 rpm</td>
<td>5</td>
</tr>
</tbody>
</table>

Fig. 3 Boundary conditions.

Fig. 4 Streamline distributions in the furnace.

Fig. 5 Axial velocity distribution for different fan speeds in the YZ-plane: (a) 1,000 rpm, (b) 2,000 rpm, and (c) 3,000 rpm.
In Fig. 6 (a) and (b), the estimated axial and tangential velocities at line 1 \((y = 0.4 \text{ m})\) are shown. These velocities increased with increasing fan speed. The maximum axial velocities were 8, 14, and 22 m/s at fan speeds of 1,000, 2,000, and 3,000 rpm, respectively. The maximum tangential velocities were estimated to be 2, 3, and 6 m/s at the three fan speeds. According to these results, swirling \(f_\text{low}\) was predicted near the 0.3 m lateral point in the lower section of the furnace. At line 2 \((y = 2.2 \text{ m})\), as seen in Fig. 6 (c) and (d), the axial velocities increased from the center of the furnace because of the pressure difference generated below the fan, and also increased with increasing fan speed. The tangential velocity reduced with the decay of the swirl in the vicinity of the fan. From these results it can be seen that when the fan speed increased, the mixing rate was increased by the velocity increasing in all directions.

To estimate the temperature uniformity induced by flow mixing in the furnace, the temperature distributions were analyzed. The temperature distribution for different fan speeds in the YZ-plane is presented in Fig. 7. The highest temperature gradient was estimated in the lower section of the furnace, because the gas inlets were located in the lower section. Also, steep temperature gradient occurs in the cover head section, but temperature variations are observed to be relatively small in the upper section of the furnace owing to the strong convection effect. Temperature uniformity improved with increasing fan speed because of the increase of convective heat transfer. Figure 8 shows the temperature profiles of mixture gas at line 1 \((y = 0.4 \text{ m})\) and line 2 \((y = 2.2 \text{ m})\). The non-uniform temperature was partially distributed at line 1, and the maximum temperature difference, based on average temperatures, was 15 K at 1,000 rpm, 6 K at 2,000 rpm, and 4 K at 3,000 rpm. In Fig. 8, it is found that the predicted gas temperature is in the range from 833 to 853 K, provided from the heat treatment industry. This shows good agreement with the process temperature re-

Fig. 6  Velocity profiles for different fan speeds: (a) axial velocity at line 1 \((y = 0.4 \text{ m})\), (b) tangential velocity at line 1 \((y = 0.4 \text{ m})\), (c) axial velocity at line 2 \((y = 2.2 \text{ m})\), and (d) tangential velocity at line 2 \((y = 2.2 \text{ m})\).

Fig. 7  Temperature distributions for different fan speeds in the YZ-plane: (a) 1,000 rpm, (b) 2,000 rpm, and (c) 3,000 rpm.
quired for surface treatment. In addition, the temperature variation was much higher near +0.2 m laterally than in the center. The reason for this asymmetric distribution at line 1 was asymmetric flow characteristics caused by the off-center location of the gas inlets and the outlet. The temperature decreased in the vicinity of the guide chamber wall at line 2 owing to heat loss at the furnace cover. Non-uniformity of mixture gas temperature and an out-of-process temperature range from 833 K to 853 K could cause a deterioration in product quality during the nitrocarburizing process. From the results we can see that, as the fan speed increased, the local temperature differences decreased in the furnace. Thus by controlling the fan speed it is possible to improve the temperature uniformity and flow mixing in the furnace.

3.2 Effect of swirling injection on thermal characteristics

As referred previously, internal flow inside the furnace can be controlled by fan speed, and it was confirmed that flow mixing and heat transfer were increased with increasing fan speed. However, for improving heat transfer, the fan speed cannot indefinitely increase because higher fan speeds lead to higher energy consumption and a resulting process efficiency drop. Therefore, this study analyzed the influence of swirling gas injection on thermal flow characteristics in the furnace. The swirl number, which is defined as the ratio of the axial flux of angular momentum to the axial flux of axial momentum, was used for controlling the swirl intensity at the gas inlet, and is defined below:

\[ S_N = \frac{\int rwv \cdot d\vec{A}}{\overline{R} \int u^2 \cdot d\vec{A}}. \]

where \( \overline{R} \) is the hydraulic radius, \( w \) is the tangential velocity, \( u \) is the axial velocity, \( \vec{v} \) is the velocity vector, and \( \vec{A} \) is the cell area. Axial- and tangential-velocity depend on the vane angle of the swirl generator. In this study, the swirl number is defined as the ratio of tangential velocity to the axial velocity with the assumption of uniform axial and tangential velocity in the pipe and constant hydraulic radius in the pipe. Five sets of swirl numbers at the gas inlet, 0, 0.5, 1, 3, and 5, were tested, and the flow mixing and heat transfer predicted at a fan speed of 1,000 rpm. The formation of the swirl flow structure is shown schematically in Fig. 9. The swirl flow from the rotational force of the fan had strong angular- and radial-momentum, and increased with height having a minimum diameter of swirl flow. The minimum diameter of swirl was predicted at about 0.4 m when fan speed was 1,000 rpm. Gas inlets were located within the minimum diameter of the swirl flow, therefore a swirling gas injection would enhance the flow mixing and heat transfer in the lower section of the furnace. Figure 10 shows the axial- and tangential-velocity profiles at line 3 (\( y = 0.1 \) m) and line 4 (\( y = 0.2 \) m). As the swirl number of the inlet gas increased, the axial velocity decreased and the tangential velocity gradually increased. The estimation of these velocities showed similar tendencies at swirl numbers 3 and 5. Tangential velocities for the different swirl numbers were predicted to have

![Fig. 8](image)

**Fig. 8** Mixture gas temperature profiles for different fan speeds: (a) Line 1 (\( y = 0.4 \) m) and (b) Line 2 (\( y = 2.2 \) m).

![Fig. 9](image)

**Fig. 9** Formation of swirl flow structure.
approximately the same value along line 4 because of internal swirling flow from the rotational force of the fan. Figure 11 shows the temperature distributions of atmosphere gases in the YZ-plane for the different swirl numbers. Temperature uniformity of inlet gas with swirl number was seen at the lower section of the furnace due to the increase in the convective heat transfer with flow mixing between hot atmosphere gas and cold inlet gas. The temperature profiles of mixture gas for line 1 ($y = 0.4$ m), line 3 ($y = 0.1$ m), and line 4 ($y = 0.2$ m) are shown in Fig. 12. The temperature increased sharply more than swirl number 1, and the swirl flow of the inlet gas could have an effect on the flow mixing and temperature distribution in the furnace. It was confirmed that the mixture gas temperature reached the process temperature range 833–853 K. From the results, swirling gas injection has a tendency to increase the flow mixing and temperature uniformity at the lower section close to the gas inlets. Moreover, regarding energy efficiency, swirling injection could promote temperature uniformity with a low fan speed.

4. Conclusions

A strong swirl flow was formed by the rotational force of the fan. A swirl flow has a tendency to enhance the flow mixing and convective heat transfer, and these parameters could be improved by controlling the swirl flow structure.

The fan plays an important role in the mixing of the internal flow in the furnace. The flow mixing rate increased with
fan speed. Flow mixing and convective heat transfer have a strong relationship. According to the flow mixing, the maximum temperature difference at the lower section decreased with increasing the fan speed. In addition, the mixture gas temperature almost reached the process temperature range (above 833 K) at a fan speed of 2,000 rpm.

The swirl flow was generated with a minimum diameter at the lower section of the furnace by strong angular- and radial-momentum. Gas inlets were located within the minimum diameter of the swirling flow. For this reason, flow mixing was weaker near the gas inlets than in other regions. Therefore, swirling gas injection could greatly affect flow mixing in the lower section of the furnace. When the inlet swirl number increased, temperature uniformity improved markedly in the lower section. In addition, the mixture gas temperature reached the process temperature at swirl numbers 3 and 5. Furthermore, the difference in temperature uniformity was predicted to be less than ±5 K for swirl numbers 3 and 5. Finally, in this study, some important design factors such as fan speed and swirling gas injection were introduced to characterize the thermal flow characteristics, and these factors would contribute to designing a furnace and determining the operating conditions.

Acknowledgement

This research was supported by the Chung-Ang University Graduate Research Scholarship in 2016 and development of human resources for design specialists in applications of functional materials.

REFERENCES

2) Holm, T. and Sporege, L. Furnace atmosphere No. 3: gas nitriding and nitrocarburizing, Linde AG.

Fig. 12 Temperature profiles for different swirl numbers: (a) line 1 (y = 0.4 m), (b) line 3 (y = 0.1 m), and (c) line 4 (y = 0.2 m).