Optimization of 3-D natural convection around the isothermal cylinder using Taguchi method

Javad. Alinejad 1,*, Javad. Abolfazli Esfahani 2

1Department of Mechanical Engineering, Sari Branch, Islamic Azad University, Sari, Iran
2Department of Mechanical Engineering, Ferdowsi University of Mashhad, Mashhad, Iran

ABSTRACT: This study discusses the application of Taguchi method in assessing minimum entropy generation and maximum heat transfer rate for natural convection in an enclosure embedded with isothermal cylinder. The simulations were planned based on Taguchi’s L25 orthogonal array with each trial performed under different conditions of position and aspect ratio (AR) of the cylinder. The thermal lattice Boltzmann based on D3Q19 methods without any turbulent submodels was purposed to simulate the flow and thermal fields. A relaxation time method with the stability constants is introduced to solve turbulent natural convection problems. Signal-to-noise ratios (S/N) analysis were carried out in order to determine the effects of process parameters and optimal factor settings. Finally, confirmation tests verified that Taguchi method achieved optimization of heat transfer rate with sufficient accuracy.

KEYWORDS: Cylinder; Lattice Boltzmann model; Natural convection; Taguchi method

INTRODUCTION

The main idea of this study is simulation of natural convection in a cavity with a heat source. This simulation is more important in industrial and technological applications including, electronic cooling [1], cooling of the reactors [2, 3], heat and mass transfer processes in cryogenic fuel and vertical storage tanks [4–9]. In these works, the regimes of convective heat transfer in closed vertical volumes were analyzed in detail for the conditions when the thermal fluxes supplied to the liquid are uniformly distributed over the bottom and lateral surfaces. The spatial and temporal structure of convection at a sine distribution of the thermal flux on the lateral wall of the vertical cylinder was presented in [10]. The mathematical modeling of unsteady regimes of natural convection in a closed cylindrical region with a heat-conducting shell of finite thickness was carried out in [11]. Numerous studies of various convective flow based on entropy generation minimization (EGM) are reported in literature [12–15]. Chatterjee and Chakraborty [12] examined the numerical formulation involving second law of thermodynamics for entropy generation analysis of three-dimensional surface tension driven turbulent transport during laser materials processing. Esfahani and Alinejad [13] analyze the entropy generation due to conjugate natural convection in an enclosure. Transition criteria for entropy reduction of convective heat transfer from micropatterned surfaces were reported by Naterer [14].

Entropy generation in microchannel flow with presence of nanosized phase change particles was investigated by Alquaity et al. [15]. Recently Lattice Boltzmann Method (LBM) has been developed as a new tool for simulating the fluid flow, heat transfer and other complicated physical phenomena. Compared with the traditional computational fluid dynamics methods, the Lattice Boltzmann Method is a meso-scale modeling method based on the particle kinematics. It has many advantages, such as simple coding, easy implementation of boundary conditions and fully parallelism. At present the applications of LBM have achieved great success in multi-phase flow, chemical reaction flow, thermal hydrodynamics, suspension particle flow and magneto hydrodynamics. Esfahani and Alinejad [16] conducted the simulation for viscous-fluid flow and conjugate heat transfer in a rectangular cavity by using LBM. D’Orazio et al. [17] performed the numerical calculations for the natural convection in a cavity. In the different study several designs of experiments (DOE) approaches have been applied to improve the efficiency of thermal system. A number of research studies have reported that the Taguchi design is an ideal method. The Taguchi approach is a simple and easy tool, which provides effective solutions on a design, as it emphasizes a mean performance value close to the target value. In this way, significant factors that have a significant impact on the experimental condition could be recognized and the optimal performance is determined [18–20]. This Paper investigates the natural convection heat transfer in a three-dimensional cavity with the presence of an isothermal cylinder as a heat transfer...
source by means of Taguchi method in order to obtain significant process parameters and optimum factor combinations. For this reason the thermal Lattice-Boltzmann method with the Boussinesq approximation is employed to simulate natural convection. The effects of cylinder aspect ratio and position has been observed and analyzed in detail. The present results provide a good approximation for choosing effective parameters of designing thermal system.

LATTICE BOLTZMANN METHOD

The lattice Boltzmann method is particularly successful as a numerical method for solving the different fluid dynamic problems [21-23]. LBM is derived from lattice gas methods as an explicit discretization of the Boltzmann equation in the phase space is considered. LBM is a vigorous numerical method, based on the kinetic theory to simulate fluid flow and heat transfer [24-25]. Unlike the classical macroscopic approach (Navier-Stokes) the lattice Boltzmann is a mesoscopic model to simulate flow field [25-26]. In this approach, the fluid domain is made discrete in uniform Cartesian cells, each one of which holds a fixed number of distribution functions (DF) that represent the number of fluid particles moving in these discrete directions. Hence depending on the dimension and number of velocity directions, there are different models that can be used. The present study examined three-dimensional flow by using 3-D lattice with nineteen velocities (D3Q19 model). The velocities of the D3Q19 model are shown in Figure. 1. The LB Model used in the present work is the same as that employed in [27-29]. The DFs are calculated by solving the Lattice Boltzmann Equation (LBE), which is a special discretization of the kinetic Boltzmann equation. After introducing Bhatnagar-Gross-Krook (BGK) approximation, the Boltzmann equation can be formulated as below [22]:

\[
f_k(x + c_k \Delta t, t + \Delta t) = f_k(x, t) + \frac{\Delta t}{\tau} \left[ f^e_k(x, t) - f_k(x, t) \right] + \Delta t c_k F_k
\]

Where \( \Delta t, c_k, \tau, f^e_k, F_k \) denote lattice time step, the discrete lattice velocity in direction \( k \), the lattice relaxation time, the equilibrium DF, and the external force in the direction of the lattice velocity, respectively.

The equilibrium distribution function for the D3Q19 velocity model is given by

\[
f^e_k(x, t) = \omega_k \rho \delta \left[ 1 + \frac{3c_k u}{c^2} + \frac{9(c_k w)^2}{2c^2} - \frac{3u^2}{c^2} \right] \tag{2}
\]

where \( \omega_0 = \frac{1}{3}, \omega_k = \frac{1}{18} \) for \( k = 1-6 \) and \( \omega_k = \frac{1}{36} \) for \( k = 7-18 \). In order to incorporate buoyancy force in the model, the force term in equation 1 needs to be calculated, as below, in a vertical direction \( z \):

\[
F = \rho g z \beta \Delta T \tag{3}
\]

The macroscopic fluid densities and velocities are computed as below:

\[
\rho = \sum_{k=0}^{18} f_k, \quad u = \frac{1}{\rho} \sum_{k=0}^{18} f_k c_k \tag{4}
\]
For the temperature field the g distribution is as below:

\[ g_k(x + c_k \Delta t, t + \Delta t) = g_k(x, t) + \frac{\Delta t}{\tau_g} [g_k^{eq}(x, t) - g_k(x, t)] \]  

(5)

For the D3Q19 model, the equilibrium energy distribution functions can be defined as follows:

\[ g_0^{eq} = -T \frac{u^2}{c^2} \]  

(6)

\[ g_{1-6}^{eq} = \frac{T}{18} \left[ 1 + \frac{c_k u}{c^2} + \frac{9(c_k u)^2}{2c^2} - \frac{3u^2}{2c^2} \right] \]  

(7)

\[ g_{7-18}^{eq} = \frac{T}{36} \left[ 2 + 4 \frac{c_k u}{c^2} + \frac{9(c_k u)^2}{2c^2} - \frac{3u^2}{2c^2} \right] \]  

(8)

The temperature field is computed as:

\[ T = \sum_{k=0}^{18} g_k \]  

(9)

### Natural convection stability constant (sc)

For stable and convergence solution in natural convection the maximum velocity should not exceed the critical magnitude (\( u_{max} \leq 0.1 \)) in terms of compressibility effect and the relaxation time must be as large as possible. For these reasons we use the scale analysis of the natural convection boundary layer to find the suitable relaxation time as follows:

For \( Pr < 1 \):

\[ u_{max} \sim \frac{\theta}{H} \left( \frac{Ra^{1/2}}{Pr} \right)^{1/2} \]  

(10)

\[ Ma = \frac{u_{max}}{c_s} \]  

(11)

Proportionalities (10) and (11) can be converted to equalities using constants sc1 and sc2. These constants could be calculated for different geometries by solving the problem in a more stable condition.

For instance, by comparing the two sides of these inequalities, we obtained sc1 = 0.25 for a 2-D convective square cavity problem with Ra = 106.

The fact that sc1, and sc2, is less than unity is an important result, which has not been mentioned in the previous works. The significance of this point would be discovered later on.

For \( Pr < 1 \):

\[ u_{max} = sc_1 \frac{\theta}{H} \left( \frac{Ra^{1/2}}{Pr} \right) \]  

(12)

\[ Pr > 1: u_{max} = \frac{\theta}{H} \left( \frac{Ra^{1/2}}{Pr} \right) \]  

Substituting \( u_{max} \leq 0.1 \) and solving for \( \theta \):

For \( Pr < 1 \):

\[ \theta \leq \frac{0.1}{sc_1} \left( \frac{1}{c_s^2} \right) \left( \frac{Pr^{1/2}}{Ra} \right) H + 0.5 \]  

(16)

\[ \tau_g \leq \frac{0.1}{sc_1} \left( \frac{1}{c_s^2} \right) \left( \frac{1}{RaPr} \right) H + 0.5 \]  

(17)

By these equations the relaxation times calculated as below:

For \( Pr > 1 \):

\[ \theta \leq \frac{0.1}{sc_2} \left( \frac{1}{c_s^2} \right) \left( \frac{Pr}{Ra^{1/2}} \right) H + 0.5 \]  

(18)

\[ \tau_g \leq \frac{0.1}{sc_2} \left( \frac{1}{c_s^2} \right) \left( \frac{1}{Ra^{1/2}} \right) H + 0.5 \]  

(19)

### Nusselt number

Heat transfer between hot and cold walls was computed by local and mean Nusselt number which is given as

\[ N_u_l = \frac{-1}{\tau_{m}} \frac{\partial \theta}{\partial n} \text{wall}, \quad \tau_{m} = \frac{T - T_c}{T_h - T_c} \]  

(20)

\[ N_u_m = \frac{1}{T} \int_0^T N_u_l \, dx \]  

(21)

### Entropy generation

\[ N_s = \left[ \left( \frac{\partial \theta}{\partial x} \right)^2 + \left( \frac{\partial \theta}{\partial y} \right)^2 \right] + \phi \left[ \left( \frac{\partial u}{\partial x} \right)^2 + \left( \frac{\partial v}{\partial y} \right)^2 \right] + \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)^2 \]  

\[ \phi = \frac{\mu T_0}{k} \left( \frac{\alpha}{H \Delta T} \right)^2 \]  

(22)

\[ S_g = \int \limits_{\gamma} N_s \, d\gamma \]  

(23)

**PHYSICAL MODEL**

The physical geometry considered in this study is shown in Figure 2. We consider the natural convection of a viscous incompressible fluid in a three dimensional enclosure in the presence of a solid local energy source with constant temperature \( T_h \) and the wall temperature of the cavity are
constant which contact to the outside with the ambient temperature ($T_c$).

When doing numerical simulation it was assumed that the thermophysical properties of the material are temperature-independent. In the present study the flow is bounded by an enclosure with the geometric setup, where $D = 2\sqrt{ab}$ and $H = 5D$ denote the cylinder diameter and the height of the cavity, respectively.

![Curved boundary treatment](image)

**Fig. 2.** Schematic diagrams of the cavity and different cylinders

**Curved boundary treatment**

Figure 3a shows the imaginary part of the curved boundary, where the black small circles ($x_w$), the open circles ($x_f$) and the grey circles ($x_b$) represent the boundary, the fluid region and the solid region nodes, respectively. To apply the boundary conditions and completion of the streaming step, the reflected distribution function $\tilde{f}(x_b, t)$ on the curved boundary must be calculated. The Distance between the boundary and the fluid nodes as a fraction of the total distance between solid and fluid nodes is calculated as follows:

$$\Delta = \frac{\|x_f - x_w\|}{\|x_f - x_b\|}$$

(24)

Figures 3(b-d) show the three type of the possible boundary between the solid and fluid nodes. In all these conditions the reflected distribution function are unknown, which must be calculated.

![Curved boundary treatment](image)

**Fig. 3.** Layout of the regularly spaced lattices and curved wall boundary. (a) overall view. (b) $\Delta = 0.5$ , (c) $\Delta < 0.5$ , (d) $\Delta > 0.5$

This method is pointed in [30]. The post-collision distribution function is conducted as

$$\tilde{f}_\alpha(x_b, t + \Delta t) = (1 - \lambda) \tilde{f}_\alpha(x_f, t + \Delta t) + \lambda \tilde{f}_a^0(x_b, t + \Delta t) - 2 \lambda \frac{3}{c^2} \omega_\alpha \rho(x_f, t + \Delta t) e_\alpha \cdot u_w$$

(25)
Where

\[ f_a(x, t + \Delta t) = \frac{1}{3} w_a(x, t + \Delta t) e_a(\bar{u}_b - \bar{u}_f) \]

\[ u_b = \bar{u}_f, \lambda = \frac{2 \Delta - 1}{\tau_m - \frac{1}{2}} \quad \text{if} \quad 0 < \Delta \leq \frac{1}{2} \]  

\[ u_b = \left(1 - \frac{3}{2 \Delta} \right) \bar{u}_f + \frac{3}{2 \Delta} w, \lambda = \frac{2 \Delta - 1}{\tau_m + \frac{1}{2}} \]

\[ \text{if} \quad \frac{1}{2} < \Delta \leq 1 \]

**Grid independency**

For grid independency, the average Nusselt number was calculated at high Ra numbers for different grid points.

As seen in table 1 for grid points passing from 80×80×80 to 100×100×100 for Ra = 106 and from 150×150×150 to 200×200×200 for 109, respectively, no considerable change in the average Nusselt number is observed (maximum variation is less than 0.3 %). According to the Table 1, the 100×100×100 grid points was used for Ra ≤ 108 and 200×200×200 grid points was used only for Ra = 109.

**Table 1**

<table>
<thead>
<tr>
<th>Mesh size</th>
<th>40×40</th>
<th>80×40</th>
<th>100×40</th>
<th>100×80</th>
<th>100×160</th>
<th>200×160</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ra= 10^6</td>
<td>8.815</td>
<td>8.843</td>
<td>8.845</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Ra= 10^9</td>
<td>-</td>
<td>-</td>
<td>52.374</td>
<td>53.116</td>
<td>53.264</td>
<td></td>
</tr>
</tbody>
</table>

**CODE VALIDATION**

The numerical simulation was done by an in-house code written in FORTRAN, using LBM. Numerical investigations were carried out for the following values of dimensionless Rayleigh number, \(10^6<\text{Ra}<10^9\). The influence of the main parameters characterizing the process was analyzed.

The obtained results are compared with the previous 2-D and 3-D simulations of natural convection in a square cavity [33-38].

The comparison of streamlines, isotherms and mean Nusselt number at the interface between the solid wall and gaseous cavity with previous work at different Rayleigh numbers illustrates a fine agreement that has been obtained (figure 4 and table 2).

The isotherm lines vortex indicates a change in the dominant heat transfer mechanism with Rayleigh number. For low Rayleigh number, isotherms are aligned with the temperature constant walls and slightly deviated by the flow, and the heat is transferred mainly by heat conduction. As Ra increases, the controlling heat transfer mechanism changes from conduction to convection, the shape of isotherms begins to bend in the bulk region. The isotherm lines become flat in the central region of the cavity. These lines are vertical only in thin boundary layers near the hot and cold walls and the fluid is thermally arranged in different layers.

In other words, the isotherms become horizontal in the cavity.

Observing the streamlines patterns reveals wavy disturbances occurrence close to the horizontal adiabatic boundary specially at upper-left and bottom right corners. These patterns intensify by increasing Ra to 10^9 and finally eddies are developed.

The temperature field becomes more and more stratified. The isotherms near the hot wall stretch upward as a result of the warm fluid wake.

Finally, It should be noted that there is an excellent agreement between the present results and the benchmark LBM 2-D solution by Du et al. [31] and Dixit et al. [32] and 3-D solution by Peng et al. [34] for all values of Ra (maximum difference is less than 8%), as well as with the CFD 2-D solutions by Barakos et al. [33] and 3-D solutions by Bocuc and Altac [35] (maximum difference is less than 4%).

**Table 2**

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>10^6</td>
<td>1.113</td>
<td>1.117</td>
<td>1.108</td>
<td>1.121</td>
<td>1.114</td>
<td>1.099</td>
<td>1.070</td>
</tr>
<tr>
<td>10^8</td>
<td>8.845</td>
<td>8.767</td>
<td>8.822</td>
<td>8.652</td>
<td>8.806</td>
<td>8.046</td>
<td>8.794</td>
</tr>
</tbody>
</table>

Figure 5 shows the variations of dimensionless temperature and the vertical velocity profiles, along the mid-height line of cavity for different Rayleigh number. It can be seen that, by increasing the Ra number these profiles begin to bend near the vertical hot and cold walls. Also, it can be found that very close to the hot wall, the temperature profiles are nearly linear and near the cold wall are almost anti-symmetric.

It can be seen that, there is a good agreement between the present prediction and the previous data.
RESULTS AND DISCUSSION

The effect of different aspect ratio

The streamlines (figure 6a) and isotherms (figure 6b) are observed in Figure 6.

Illustrative figures are shown at $Ra=10^5$ and different AR (Aspect Ratio, $AR=a/b$) in various cases.

As a result of the buoyancy effect, the fluid from the heat source in the cavity rises and flow down along the vertical walls forming two main rolls in the cavity. A detailed examination of the streamlines and isotherms clearly shows that the cold fluid is entrained towards the heated cylinder which, in turn, rises upward there by setting up a buoyancy induced flow which transports heat from the heated cylinder to the cold ambient.

In addition, this phenomenon gives rise to the formation of two main circulations.

The main circular cells occupy a region between left and right vertical wall. The effect of the shape of the cylinder cross section (aspect ratio $AR=a/b$) is also seen to be significant.

For instance, in the case of the so-called slender configuration ($AR<1$), the plume was seen to be smaller than that for a circular cylinder ($AR=1$) otherwise under identical conditions. Of course, the reverse trend is seen for blunt configuration ($AR>1$).
Application of the Taguchi method

Taguchi method is proposed by Taguchi in 1960s. This method is widely applied for improving industrial product quality greatly [36-37]. In addition to this, low trial numbers, obtaining the effects of process parameters on quality characteristics and their optimum levels has easily increased its popularity.

In the present study, the effects of aspect ratio and position of the cylinder on heat transfer rate (Nu) and entropy generation (Sg) have been determined and optimum factor levels have been obtained by analyzing Taguchi method.

The number of simulation can be reduced by means of Taguchi technique. Based on orthogonal arrays. This method uses the special design of orthogonal arrays to learn the whole parameters space with small number of experiments.

In the present study, an L25 orthogonal array by three factors with five levels is chosen as shown in Table 3. To get more accurate in terms of the heat transfer rate, the Nusselt numbers over the wall of the cavity and entropy generation number in flow field for various designated trial have been calculated.

Taguchi method employs a signal-to-noise ratio (S/N) to measure the present variation.

Also, in calculation procedure the effect of different control factors and their interaction was assumed. In Taguchi designs, a measure of robustness used to identify control factors that reduce variability in a product or process by minimizing the effects of uncontrollable factors (noise factors).

Higher values of the signal-to-noise ratio (S/N) identify control factor setting that minimize the effects of the noise factors.

The definition of S/N ratio differs according to an objective function, i.e., a characteristic value.

There are three kinds of characteristic value: Nominal is Best (NB: \( n = 10 \log_{10} \text{square of mean/variation} \)), Smaller is Better (SB: \( n = -10 \log_{10} \text{mean of sum of squares of measured data} \)) and Larger is Better (LB: \( n = -10 \log_{10} \text{mean of sum squares of reciprocal of measured data} \)). As the minimum entropy generation (Sg) is the one of the major goal in present study, SB is chosen as a characteristic value. S/N ratios plots for different factors are given in Figure 7.

As a result, the optimum settings of control factors minimizing the entropy generation are AR=2, r=0.7 R and \( \phi=135^\circ \).

### Table 3

<table>
<thead>
<tr>
<th>Aspect Ratio</th>
<th>Radius (r)</th>
<th>Angle (( \phi ))</th>
<th>Sg</th>
<th>Nu</th>
</tr>
</thead>
<tbody>
<tr>
<td>AR = 1/2</td>
<td>0</td>
<td>0(^\circ)</td>
<td>280</td>
<td>1.906</td>
</tr>
<tr>
<td></td>
<td>0.3 R</td>
<td>45(^\circ)</td>
<td>339</td>
<td>1.807</td>
</tr>
<tr>
<td></td>
<td>0.5 R</td>
<td>90(^\circ)</td>
<td>211</td>
<td>1.751</td>
</tr>
<tr>
<td></td>
<td>0.6 R</td>
<td>135(^\circ)</td>
<td>73</td>
<td>1.929</td>
</tr>
<tr>
<td></td>
<td>0.7 R</td>
<td>180(^\circ)</td>
<td>84</td>
<td>2.884</td>
</tr>
<tr>
<td>AR = 2/3</td>
<td>0</td>
<td>45(^\circ)</td>
<td>273</td>
<td>1.858</td>
</tr>
<tr>
<td></td>
<td>0.3 R</td>
<td>90(^\circ)</td>
<td>255</td>
<td>1.793</td>
</tr>
<tr>
<td></td>
<td>0.5 R</td>
<td>135(^\circ)</td>
<td>93</td>
<td>1.645</td>
</tr>
<tr>
<td></td>
<td>0.6 R</td>
<td>180(^\circ)</td>
<td>99</td>
<td>1.839</td>
</tr>
<tr>
<td></td>
<td>0.7 R</td>
<td>0(^\circ)</td>
<td>349</td>
<td>1.442</td>
</tr>
<tr>
<td>AR = 1</td>
<td>0</td>
<td>135(^\circ)</td>
<td>262</td>
<td>1.788</td>
</tr>
<tr>
<td></td>
<td>0.3 R</td>
<td>135(^\circ)</td>
<td>158</td>
<td>1.630</td>
</tr>
<tr>
<td></td>
<td>0.5 R</td>
<td>180(^\circ)</td>
<td>112</td>
<td>1.629</td>
</tr>
<tr>
<td></td>
<td>0.6 R</td>
<td>0(^\circ)</td>
<td>354</td>
<td>1.524</td>
</tr>
<tr>
<td></td>
<td>0.7 R</td>
<td>45(^\circ)</td>
<td>136</td>
<td>1.665</td>
</tr>
<tr>
<td>AR = 3/2</td>
<td>0</td>
<td>135(^\circ)</td>
<td>256</td>
<td>1.756</td>
</tr>
<tr>
<td></td>
<td>0.3 R</td>
<td>180(^\circ)</td>
<td>162</td>
<td>1.635</td>
</tr>
<tr>
<td></td>
<td>0.5 R</td>
<td>0(^\circ)</td>
<td>353</td>
<td>1.604</td>
</tr>
<tr>
<td></td>
<td>0.6 R</td>
<td>45(^\circ)</td>
<td>220</td>
<td>1.544</td>
</tr>
<tr>
<td></td>
<td>0.7 R</td>
<td>90(^\circ)</td>
<td>150</td>
<td>2.370</td>
</tr>
<tr>
<td>AR = 2</td>
<td>0</td>
<td>180(^\circ)</td>
<td>254</td>
<td>1.748</td>
</tr>
<tr>
<td></td>
<td>0.3 R</td>
<td>0(^\circ)</td>
<td>329</td>
<td>1.721</td>
</tr>
<tr>
<td></td>
<td>0.5 R</td>
<td>45(^\circ)</td>
<td>241</td>
<td>1.531</td>
</tr>
<tr>
<td></td>
<td>0.6 R</td>
<td>90(^\circ)</td>
<td>167</td>
<td>1.911</td>
</tr>
<tr>
<td></td>
<td>0.7 R</td>
<td>135(^\circ)</td>
<td>40</td>
<td>3.005</td>
</tr>
</tbody>
</table>
The second objective function is the maximum heat transfer rate (Nu), so LB is chosen as a characteristic value. S/N ratios plots for different factors are given in Figure 8. As a result, the optimum settings of control factors maximizing the Nusselt number are AR=1/2, r=0.7 R and φ=135°.

Finally the combination of the maximum heat transfer rate and minimum entropy generation is chosen as a major objective function. S/N ratios plots for different factors are given in Figure 9.

As a result, the optimum settings of control factors maximizing the Nusselt number and minimizing the entropy generation are AR=1/2, r=0.7 R and φ=135°. The final step in verifying results based on Taguchi design is the confirmation test.

Figure 10 shows the streamlines and isotherms for optimal factor settings. Numerical results indicate that the Nusselt number and entropy generation in this case are equal to 3.05 and 54, respectively.

It can be concluded that Taguchi method achieves the Statistical assessment of maximum heat transfer rate and minimum entropy generation of natural convection in an enclosure embedded with isothermal cylinder.
CONCLUSION

In this article, effect and optimization of different parameters in natural convection in a three-dimensional cavity with the presence of a hot cylinder on maximum Nusselt number and minimum entropy generation were investigated through Taguchi method. The practical benefit of this study is that the use of obtained optimum condition increases the heat transfer rate and decreases the entropy generation. The simulation is numerically predicted by using Lattice Boltzmann Method. In conclusion, some of the main points are briefly remarked:

- In comparison with complicated model such as large-eddy simulation or MRT, using the mentioned technique to simulate turbulent natural convection in the present study having a simple calculation procedure.
- Comparisons of the results with previous work at high Rayleigh numbers show that a very good agreement has been obtained.
- Based on the SN ratio plots, all control factors have significant effect on the quality characteristic statistically.
- The optimum condition with the maximum Nusselt number was obtained on the fourth level of radius \((r=0.7 \, R)\), the third level of angle \((\phi =135^\circ)\) and the first level of aspect ratio \((\text{AR}=1/2)\).
- The optimum condition with the minimum entropy generation was obtained on the fourth level of radius \((r=0.7 \, R)\), the third level of angle \((\phi =135^\circ)\) and the fifth level of aspect ratio \((\text{AR}=2)\).
- The optimum condition with the combination of the maximum heat transfer rate and minimum entropy generation was obtained on the fourth level of radius \((r=0.7 \, R)\), the third level of angle \((\phi =135^\circ)\) and the first level of aspect ratio \((\text{AR}=1/2)\).
- The confirmation test including optimum condition was conducted and the obtained result indicates that Taguchi method can be used in the optimization of mix convection cooling successfully.

REFERENCES


