Simulation of water-circulating path and heat transfer in blast-furnace tuyeres with simple and spiral double-chamfer design

Si. Parchami Borjoui Raky [1], So. Parchami Borjoui Raky [2], M. Sharifi *[3], S. Mardani [4]

Research and Development Unit of Parchamdaran Sanat Sepahan company, Isfahan, Iran

Abstract

Tuyeres or air-blowing nozzles are considered sensitive parts of blast furnaces responsible for blowing hot air into the furnace, and their proper operation has a direct effect on the iron-melting process in the blast furnace. These parts are usually damaged or destroyed by exposure to high temperatures, radiation, slag or molten iron scattering, improper design of the water-circulating system, low thermal conductivity, operator problems, etc. In this paper, the two simple and spiral water-circulating designs are simulated and examined in heat transfer and fluid pressure drop under furnace conditions. Based on the results obtained from the spiral design, water is directed to the critical areas of the body at a lower temperature and passes through these areas quicker than the simple design.; Additionally, the cooling power of the spiral design is more efficient for the body of the tuyeres and will help prevent this part from burning in these areas. According to the simulation, the spiral design's water pressure will drop slightly further than the simple design.

Keywords: Tuyeres, Air-blowing nozzle, Blast furnace, Spiral water-circulating design, Heat transfer.

1. Introduction

Restricted water and energy resources to be used in heavy industries are among the challenges facing Iran's economy and the world. Moreover, due to the old designs existent in the industry, the importance of a holistic scientific view is inevitable. Iron and steel manufacturing are amongst significant heavy industries consuming a considerable amount of energy and water resources. Furnaces have extremely high internal temperatures due to the molten material inside. Since the production of steel requires the continuous operation of the furnaces, and any technical problems or discontinuation in the production line, reduce fabrication capacity and cause significant loss to the relevant industrial unit, the various components of the production line must be optimally designed and resistant to high temperatures and thermal stresses [1)].

One of the essential types of equipment in blast furnaces are tuyeres, which are responsible for blowing hot air (1100-1250 °C) and charging materials for a timely ignition of coke while performing chemical reactions between metals in an area of the furnace with the highest temperature [2-3)]. Since this equipment is installed and used in a part of furnaces with a temperature range of 1600-2500 °C (Fig. 1), its surface temperature control (up to a maximum temperature of 600 °C) is of great importance

---

*Corresponding author
Email:msharifi1991@yahoo.com
Address: Research and Development Unit of Parchamdaran Sanat Sepahan company, Isfahan, Iran
1. B.Sc
2. M.Sc
3. M.Sc
4. B.Sc*

to prevent premature destruction of the tuyere and to enable proper operation [2-4]. Cold water flows inside the tuyeres to support them against the internal heat of the furnace. The tuyere behaves like a heat exchanger, with air acting as hot fluid and water as cold fluid.

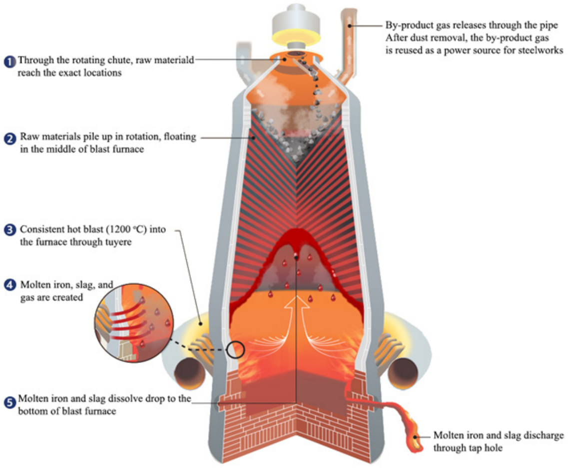


Fig. 1. Schematic image of a blast furnace.

The tuyeres used in blast furnaces are made of copper and cooled by water -circulation systems with different designs. The following is a brief example of some of the factors destroying tuyeres [5]:

1. Direct damage: Direct contact with high-temperature materials such as molten metal, slag, and coke residues.
2. Erosion due to metallurgical processes: penetration of iron in the structure of uncoated tuyeres (clad or ceramic) and consequently heat transfer reduction of copper and destruction of the part.
3. Improper design of the water -circulation system.
4. Defects in the tuyere production process: including defects in casting and welding that involve the heterogeneous structure of the part.
5. The insufficient number of tuyeres applied in blast furnaces is based on their dimensions and capacities.

The predominant technology in Iranian steelmaking is the usage of electric- arc furnaces, producing extremely high temperatures. One of the essential equipment items in furnaces is applying a suitable water -circulation system to, increase productivity significantly. One of the crucial methods to find such an optimal system is to use theoretical calculations and computer simulations. Studies have been conducted to design water -circulation and cooling systems in steel industry furnaces. Experimental and numerical simulations were carried out in the research. The following are some of these studies:

Mehrjerdi et al. [6] in 2008 simulated the thermal radiation model of the water -circulation panels of electric- arc furnaces at Mobarakeh Steel Complex. In their research, the thermal analysis of the radiation model was designed with Ansys software, and the fluid model was designed with Fluent and Gambit software. In 2009, Behbahani Nejad and Haji Doloo [7] investigated the improvement of the cap panels cooling system in melting furnaces 5 and 6 at Khuzestan Steel Complex. In 2013, Haji Doloo et al. [8] investigated the effect of reducing the cooling water flow rate on the temperature in the electric-arc furnace water -circulation panel at Khuzestan Steel Company. In 2014, Gharib Membani et al. [9] simulated transient heat transfer in the cooling panel of the electric-arc furnace cap in Khuzestan Steel Company. In 2017, Tabatabai et al. [10] studied the numerical heat transfer in the circulation panels of the electric- arc furnace walls using COMSOL software. In 2017, Haji Doloo et al. [11] simulated heat transfer and temperature distribution in the tube thickness of an arc electric furnace water-circulating panel using Fluent software.

A few studies have been carried out in heat transfer simulation, design, and optimization of water -circulation systems in the Iranian steel industry, mainly related to the water -circulation panels of arc furnaces. Little attention has been paid to the simulation and optimal design of tuyeres. In this research, two different designs of water-circulation systems related to the blast furnace tuyere No. 3 in Isfahan Steel Company (here after referred to as the furnace tuyere 3) will be investigated for fluids and heat transfer using the Multiphysics COMSOL software. In this furnace, 26 copper tuyeres blow air at 1250 °C produced in the air-preheater with a pressure of 3.5 bar into the furnace.

2. Experimental Procedure

2.1. Water-circulating path geometry and analysis process:

There are two front and rear water-circulating paths in the main design of the furnace's tuyere, (Fig. 2 (a)), responsible for cooling the nose-cone and the outer wall of the part, respectively. In the new design, due to the damage to the lower part of the outer wall resulting from being located on top of the molten and exposed to more radiation and heat, the spiral water-circulating system is implemented onto the rear wall (Fig. 2 (b)).

In this study, COMSOL Multiphysics software was used to simulate and compare the cooling system of the two designs mentioned above. At input, the pressure and velocity boundary conditions are used as the known variables, and at the output, the pressure boundary condition is applied as an unknown variable and is obtained by the software. The input parameters considered for analysis are extracted from the blast furnace monitoring system based on industrial designs, working conditions, and collected documents. The results compare the mentioned water -circulation systems in temperature distribution, hot and sensitive areas, water velocity, and water flow path.

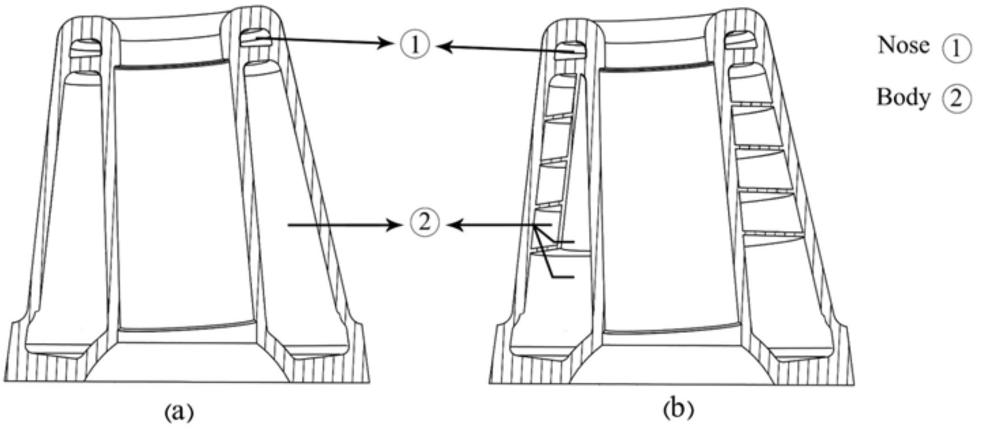


Fig. 2. Design of the furnace tuyere, a) simple design of blast furnace tuyere, b) spiral design of blast furnace tuyere

2.2. The governing equations of the problem

The finite element method is used in this simulation, and the volume, instantaneous flow rate of the fluid, and energy equations are also considered turbulent based on the fluid flow model. The kϵ model is used to simulate the turbulent flow. The main equations governing the problem are mass conservation, momentum conservation, and energy conservation, which are typical as follows:

$$\nabla.(\rho V) = 0 \qquad \text{Eq. (1)}$$

$$\rho \frac{DV}{Dt} = -\nabla P + \mu \nabla^2 V \qquad \text{Eq. (2)}$$

$$\rho C_p \frac{DT}{Dt} - k\nabla^2 T = Q \qquad \text{Eq. (3)}$$

In the above relations, heat transfer in fluid and solid is considered in the form of Fourier's law, in which μ is fluid viscosity, ρ is the fluid density, V is the fluid velocity, Q is heat flux, C_p is specific fluid heat, k is heat transfer coefficient, T is temperature and where t is time.

Because the flow regime is turbulent due to the high Reynolds number, equations (1), (2) and (3) change into relations (4), (5), and (6).

$$\rho \frac{\partial \bar{u}_i}{\partial x_i} = 0 \qquad \text{Eq. (4)}$$

$$\rho \frac{\partial \bar{u}_i}{\partial t} + \rho \frac{\partial (\bar{u}_i \bar{u}_j)}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right)\right] +$$

$$\frac{\partial}{\partial x_j}\left(-\overline{\rho \acute{u}_\imath \acute{u}_\jmath}\right) \qquad \text{Eq. (5)}$$

$$\frac{\partial \bar{T}}{\partial t} + \bar{u}_\imath \frac{\partial \bar{T}}{\partial x_i} = \alpha \nabla^2 \bar{T} - \frac{\partial (\overline{\acute{u}\acute{T}})}{\partial x} - \frac{\partial (\overline{\acute{v}\acute{T}})}{\partial y} - \frac{\partial (\overline{\acute{w}\acute{T}})}{\partial z} \qquad \text{Eq. (6)}$$

The inertial tensor $\overline{\acute{u}_\imath \acute{u}_\jmath}$ in equation (5) complicates the equation. This tensor in turbulent currents can not be ignored. The components of the tensor are not only related to the physical properties of the fluid , but also the flow conditions. Boussinesq's hypothesis relates these stresses to the mean flow velocity according to equation (7).

$$\overline{-\rho \acute{u}_\imath \acute{u}_\jmath} = \mu_t \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right) - \frac{2}{3}(\rho k + \mu_t \frac{\partial \overline{u_\imath}}{\partial \overline{x_\imath}})\delta_{ij} \qquad \text{Eq. (7)}$$

For solving turbulence equations, a mathematical model for turbulent oscillating stresses is needed. For the simulation of the turbulent flow, the standard kϵ model is used. In this model, the kinetic energy of perturbation and the rate of perturbation depreciation is obtained from the following equations.

$$\rho \frac{\partial (\rho k)}{\partial t} + \rho \frac{\partial (\rho k \bar{u}_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G + B - \rho\varepsilon$$

Eq.(8)

$$\rho \frac{\partial (\rho \varepsilon)}{\partial t} + \rho \frac{\partial (\rho \varepsilon \bar{u}_\imath)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_1 \frac{\varepsilon}{k} G +$$

Eq.(9)

$$C_1(1 - C_3)\frac{\varepsilon}{k}B - C_2 \rho \frac{\varepsilon^2}{k}$$

According to the flow rate and the difference in cooling water temperature, the heat absorbed by the water is obtained from equation (10).

$$q = \dot{m} C_p (T_{out} - T_{in}) \qquad \text{Eq. (10)}$$

The Bernoulli modified correlation was used to investigate the pressure drop, in which hf indicates the sum of the significant and minor drops:

$$\frac{P_1}{\rho} + \frac{{V_1}^2}{2} + gZ_1 = \frac{P_2}{\rho} + \frac{{V_2}^2}{2} + gZ_2 + h_f \qquad \text{Eq. (11)}$$

P_1 is the input pressure, V_1 is the input velocity, P_2 is the output pressure, and V_2 is the output velocity.

2.3. Boundary conditions and physical parameters of the problem

Given the order of the equation, solving differential equations with partial derivatives requires the initial condition and boundary conditions. The purpose of determining boundary conditions in computational fluid dynamics is to bind the discrete form of equations to solve it in a specific framework and define the flow characteristic within the computational amplitude boundaries. The boundary conditions governing the problem are used in the software, according to Table 1. In Table 2, the specifications and kind of the materials used in the simulation are shown. Thermal radiation from the furnace is also modeled to investigate heat transfer.

Table 1. Boundary condition values.

The flow rate of hot gases entering the nozzle of the tuyere	7031 m^3/h
The temperature of the hot gases at the inlet of the tuyere nozzle	1473 k
Coolant water flow rate perature	27 m^3/h
Coolant water tem	300 k
Water convective heat transfer coefficient	850 $w/(m^2.k)$
Air convective heat transfer coefficient	180 $w/(m^2.k)$
Ambient temperature	2073 k

Table 2. Material information.

Thermal conductivity of metal	401 w/(m.k)
Specific heat of metal	385 J/(kg.k)
Metal density	8940 kg/m^3
Specific heat of the coolant fluid	4181.3 J/(kg.k)
Specific heat of the fluid-cooled	1210 J/(kg.k)

2.4. Mesh generation

For converting differential equations with partial derivatives to algebraic equations, it is necessary to create a mesh. Three meshes have been selected to examine the independence of mesh dependence. Information about computing meshes is shown in Table 3. As the results of the medium and fine meshes are entirely consistent, the medium mesh is applied in all simulations.

Table 3. Information about the mesh.

Computational network	Quality criteria	The average quality of elements	Number of elements
Coarse	Elongation of elements	0.183	2154341
	Maximum angle	0.197	
Medium	Elongation of elements	0.221	5417820
	Maximum angle	0.234	
Fine	Elongation of elements	0.901	11047879
	Maximum angle	0.922	

The mesh of the computational domain is shown in Fig. 3. The type of applied elements includes quadrilateral, triangular, marginal, and vertices elements, with the shape function being second-order Lagrangian.

3. Results and discussion:

3.1. Investigation and comparison of fluid pressure drop

Fig. 4 indicates the water pressure drop in the two simple and spiral designs of the water-circulating system. According to the simulation, the pressure drop in the spiral system is more significant due to the increment in the length of the water path, increasing the main drop, and the curved path per se increases secondary drops.

3.2. Investigation and comparison of the body heat transfer of tuyeres

Fig. 5 shows the temperature of the part in two simple and spiral designs. The simulation indicates that in the spiral design, as a result of the conduction of water with lower temperature in more critical areas (, including the lower part of the tuyere body and close to the nose-cone), , which is in the higher temperature range and exposed to molten radiation, the body temperature is reduced by nearly 100 °C compared to the simple design, that is the consequence of the targeted conductivity of water.

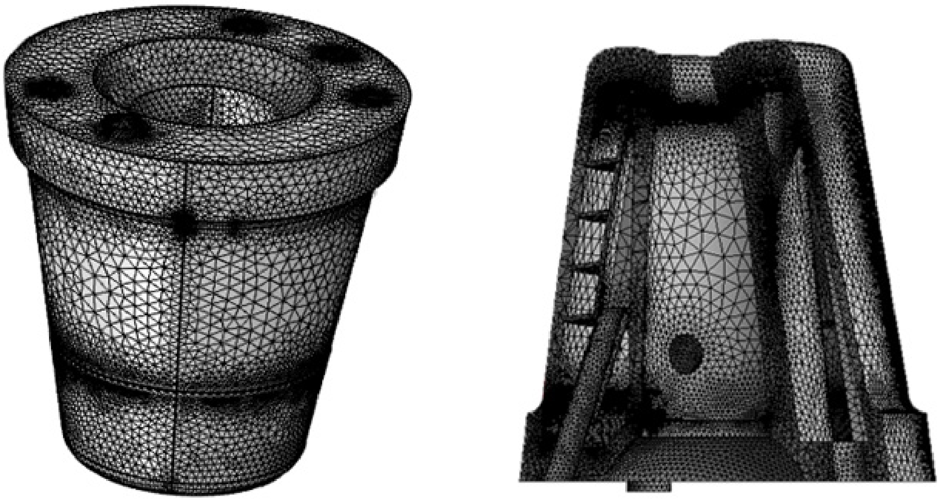
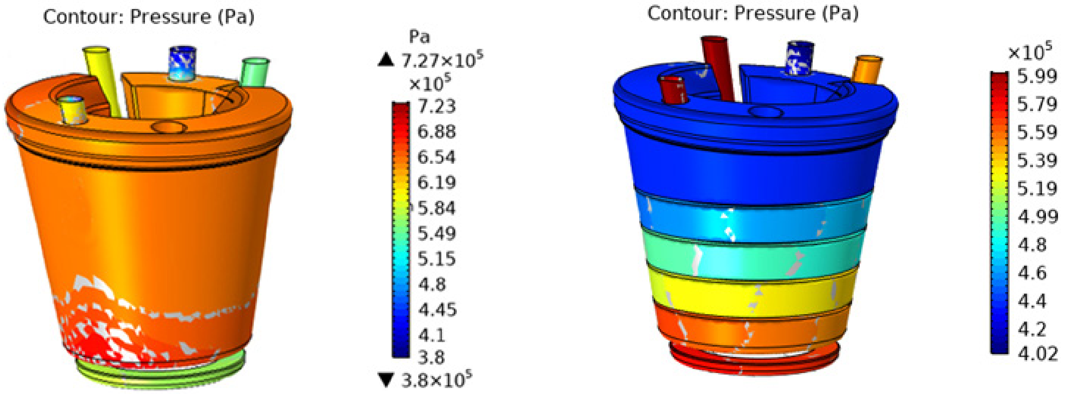
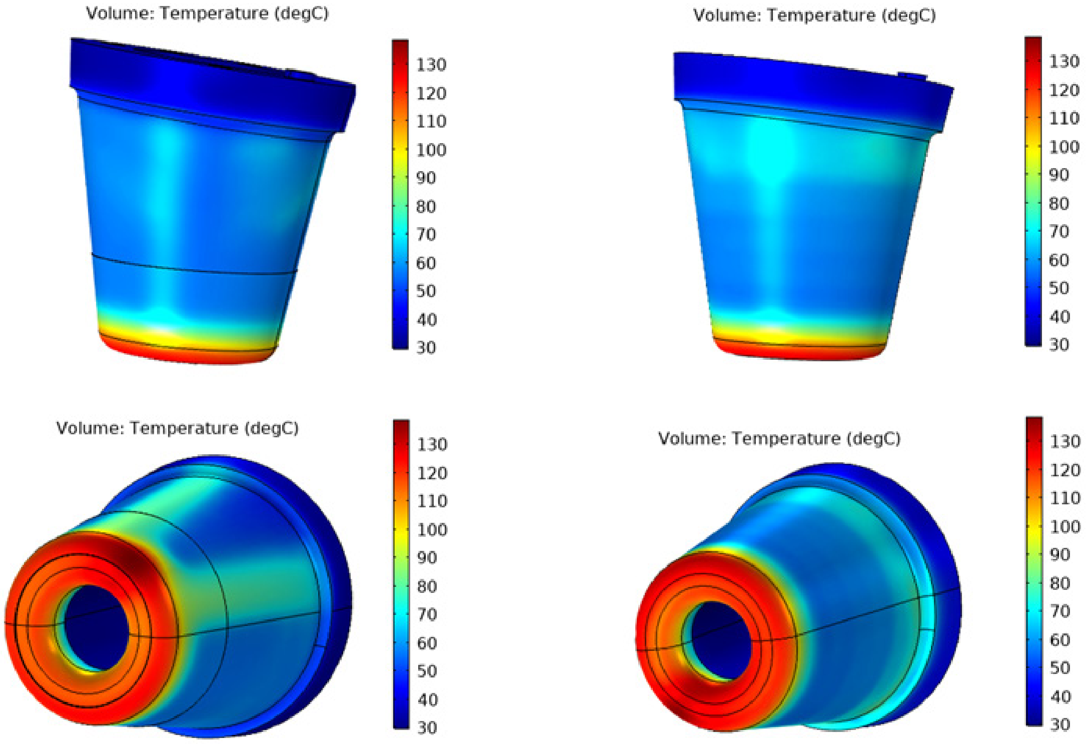


Fig. 3. The mesh of the computational domain.



(a) (b)

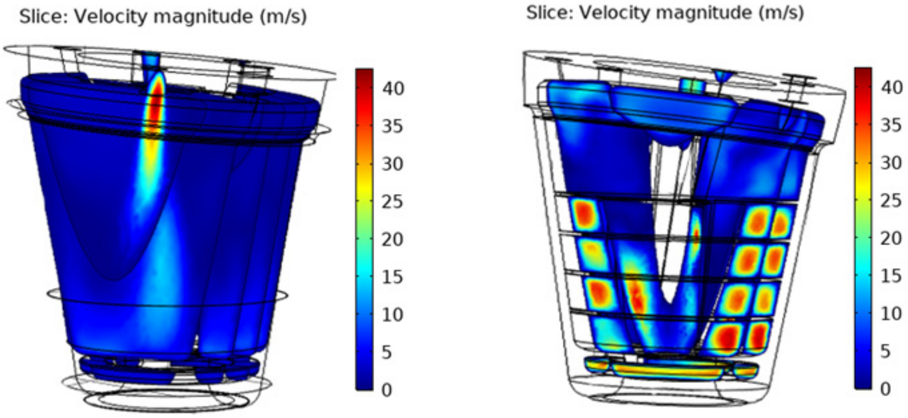
Fig. 4. Water pressure drop in the two simple and spiral designs of the water -circulation system, a) water pressure variations in a simple water -circulationg design, b)water pressure variations in a spiral water circulation design.



(a) (b)

Fig. 5. Temperature of the part in two simple and spiral designs, a) body and nose-cone temperatures in a simple water-circulating system, b) body and nose-cone temperatures in a spiral water circulation system.

Based on the entry of water into the circulation system within the spiral design , and the outset of the circulating path near the nose-cone, there will be an increase in velocity in the circulating paths, which raises the Reynolds number and intensifies the heat transfer coefficient. This process helps to control water temperature in critical areas of the body. Fig. 6 demonstrates the velocity of water movement in the water -circulation system with the two simple and spiral designs.



(a) (b)

Fig. 6. Velocity of water movement in the water-circulating system of two simple and spiral designs, a) the velocity of water movement in the body of a simple water circulation-system, b) the velocity of water movement in the body of a spiral water -circulation system.

It is noteworthy that for testing and field comparison of the research, a spiral tuyere sample has been experimentally installed in blast furnace No. 3 and is currently in operation.

4. Conclusion:

- The pressure drop in the spiral system is more significant, resulting from an increase in primary and secondary drops.
- Due to the direction of water to the critical areas in the spiral design, the body temperature is reduced by 100 °C, helping prevent the tuyeres burning in this area.
- Owing to an increase in water velocity within critical areas of the body in the spiral water-circulating design, the cooling power will be more optimal in these areas.

References:

[1] M. Saremi, K. Shirvani, M. A. Shahriari, H. Bahman: Isfahan University of Technology, Steel Symposium, (1996).(in Persian)

[2] D. Kumar Rai, Dr. M. Kumar Pal: IJEMR, 7 (2017), 252.

[3] Y. Cheng-Peng, D. Shan-Wen, T. Chien-Hsiung, Y. Ruey-Jen: Energy, 42(2012), 233.

[4] S. Yuan-Sheng, L. Hou-Guo, G. Zhong-Quan, Y. Hao: AJEAS, 6(2013), 161.

[5] O. Farkas, R. Móger: STEEL RES INT, 84(2013), 1171.

[6] F. Mehrjerdi, M. Meratian, H. Idris, B. Zamani: Ahvaz, Shahid Chamran University, Steel Symposium (2008). (in Persian)

[7]M. Behbahani Nejad, E. Haji Doloo: Khuzestan Steel Complex, Shahid Chamran University, (2009). (in Persian)

[8] E. Haji Doloo, A. Gharib Membini, M. Behbahaninejad: Second National Conference On Mechanical Systems and Industrial Innovations, Ahvaz,Iran, (2013),294 (in Persian)

[9] A. Gharib Membani, E. Haji Doloo, E. Behbahaninejad: 22nd Annual International Conference on Mechanical Engineering, Ahvaz,Iran (2014),388 (in Persian)

[10] S.A.Tabatabai, S. Behrang, M.R. Naghavi: University of Tehran, Steel Symposium (2017). (in Persian)

[11] E. Haji Doloo, M. Rezaei: 25th Annual International Conference on Mechanical Engineering, Ahvaz, Iran (2017),175 (in Persian)