www.dergipark.gov.tr
ISSN:2148-3736

El-Cezerî Journal of Science and Engineering
Vol: 10, No: 1, 2023 (147-159)
DOI :10.31202/ecjse.1175261

ECJSE

Research Paper

Numerical Investigation of Cooling an Industrial Roller by Using Swirling Jets

Mustafa KILIC[1], Mahir SAHIN*[2], Mehmood IQBAL[3], Tolga DEMIRCAN[4], Zülfikar KILINC[5] Atta ULLAH[6]

[1,2*]Adana Alparslan Türkeş Bilim ve Teknoloji University, Engineering Faculty, Mechanical Engineering Department, Adana
[4]Kırıkkale University, Engineering and Architecture, Department of Mechanical Engineering Faculty, 71450 Yahşihan/Kırıkkale
[5]BOMAK Engineering and Contracting Limited Company, Adana
[3,6]Department of Chemical Engineering, Pakistan Institute of Engineering and Applied Sciences, PO Nilore
msahin@atu.edu.tr

Received: 15.09.2022 **Accepted:** 12.12.2022

Abstract: Effective cooling of industrial rollers has prime importance to prevent the quality degradation of the system and product. High temperature difference on the roller surface may result in thermal stresses and can cause deformations on roller surface and product. In order to prevent these deformations, cooling of an industrial roller by using swirling jets is investigated for different parameters numerically in this study. Effects of Reynolds number, surface heat flux and variation in inlet temperature of the fluid on the performance of an industrial roller are investigated in terms of temperature difference between inner and outer surface of the roller. ANSYS Fluent CFD program is used to simulate heat transfer and fluid flow in this numerical study. As a result, it is obtained that increasing Re number from 1000 to 1700 causes a decrease of 45.4% in the temperature difference between inner and outer surface of the roller. Increasing surface heat flux from 5000 to 12500 W/m^2 has resulted in an increase of 149.4% in difference between inner and outer surface temperature. Increasing coolant fluid inlet temperature from 5 to 20°C has resulted in an increase of surface temperature but there is no significant change in heat transfer characteristics of the system. It is evaluated that the results of this study will contribute to design more effective cooled industrial roller.

Keywords: Industrial roller, Swirling jets, Cooling, Computational Fluid Dynamics.

1. Introduction

Various types of rollers such as drive rollers, guide rollers and conveyor rollers are used in industry. Mostly industrial rollers are used in coating, drying, annealing, heat treating and metal processing. As thermal loads on industrial devices are increasing day by day with technology development and it this expense of thermal load strongly influence industrial rollers. For this reason, many studies have been done to solve the thermal stress problem and improve heat transfer performance. In literature, different methods are being used to enhance heat transfer for industrial applications. Liu et al. [1] studied the heat transfer on rotating cylinder using water spraying method for stabilization time, nozzle position, rotation speed, tilt angle and body material1. It was found that optimal position of the nozzle was directly above the cylinder and with increase in rotation speed, the temperature distribution on the surface became homogeneous. Moreover, the selection of steel material had shown a better heat transfer effect. Lu et al. [2] investigated the effect of thermal stresses on the cylinder surface numerically by using rotating impinging jets for different Reynolds numbers, rotational velocities and heat transfer using different types of fluids (Automatic Transmission Fluid (ATF), isobutyl-alcohol, water, acetone). The use of ATF fluid provided the

How to cite this article
Kilic,, M, Sahin, M.,Iqbal,M., Demircan T., Kilinc, Z., Ullah, A. "*Numerical Investigation of Cooling an Industrial Roller by Using Swirling Jets*" El-Cezerî Journal of Science and Engineering, 2023, 10(1); 147-159.
ORCID ID: [1]0000-0002-8006-149X; [2]0000-0002-9565-9160, [3]0000-0003-1509-1163, [4]0000-0003-4805-6428, [5]0000-0003-4270-712X, [6]0000-0001-8010-3904

most optimal heat transfer, and the surface temperature decreased due to the increase in the rotational speed and the Reynolds number.

Luo et al. [3] numerically analysed the heat transfer effect of the cylinder surface and the hydrodynamics of the droplets using the multiple droplets successively impacting on cylindrical surface. They found out that reduction of vertical gap between the droplets increased the heat transfer effects on the surface. Du et al. [4] evaluated the heat transfer effect of rotation number, rotation direction and density ratio on vortex cooling under rotating conditions. It was perceived that an increase in rotation number resulted in decrease in speed and pressure and the direction of rotation generally did not have any effect on vortex cooling. Moreover, as the density ratio increased, the heat transfer density increased. Zhao et al. [5] performed spray cooling method to investigate the effects of heat flux, sub-cooling, nozzle-to-surface height, and injection pressures on surface temperature. It was observed that the spray volumetric flux was an effective parameter for this experiment. Modak et al. [6] assessed the heat transfer characteristics of CuO–H_2O nanofluid jet on a hot surface for different Reynolds numbers (5000<Re<12,000) of nanofluid having concentrations of 0.15% and 0.6% and two different nozzles to plate distance (l/d=6, 12)6. When the nanofluid concentration increased from 0.15% to 0.60%, the Nusselt number was determined as 14% and 62% respectively. Furthermore, it was shown that nanofluids had better heat transfer effect than pure water.

Selimefendigil et al. [7] examined nanofluid jet impinging cooling of an isothermal hot surface with an adiabatically rotating cylinder. In this perspective, Reynolds number (between 100 and 400), the angular rotation speed of the cylinder (between -0.1 to 0.1), the horizontal position of the cylinder (between 0 to 3.75 W) and solid particle volume fraction (between 0 and 0.04) were studied numerically. It was deduced that highest volume fraction enhanced the Nusselt number by 8.08% and heat transfer is increased with increase in the Reynolds number. Mahdavi et al. [8] studied the effects of air/nanofluid jet cooling flow on heat transfer on a hot circular rotating disc based on parameters like volume fractions, disc rotation, and Reynolds numbers. It was inferred that by increasing the angular velocity and nanoparticles concentration, the Nusselt number enhanced too. Baghel et al. [9] evaluated the heat transfer effect of a heated semi-cylindrical convex curved and inclined liquid jet impact on flat surfaces for the jet plate spacing, inclination angles and Reynolds numbers experimentally. It was observed that thermal effect of the cylindrical surface improved at high angle of inclination and high Reynolds numbers. Pachpute et al. [10] studied the heat transfer effect of impinging jets on a heated cylinder experimentally. Moreover, Reynolds number (Re=5000-20000), curvature ratio (5.1-20.4), nozzle-cylinder spacing (2-12) and thermal effects on the cylinder were analyzed. No rise in Nusselt number was observed when the number of jets increased.

Jahedi et al. [11] studied the cooling rate of rotary hollow cylinder by one row of water impinging jets using jets range, different Reynolds number, rotation speed, sub-cooling, and impingement impact angle of jets. An increase in temperature drop above 400 degrees was observed at high Reynolds numbers and a more homogeneous heat transfer effect was detected at low rotation rates. Patil et al. [12] conferred the heat transfer effect of a row of circular jets impinging on a concave surface. They also studied the thermal effect for the nozzle length diameter ratio, nozzle target spacing and different Reynolds numbers. Jiang et al. [13] investigated the thermal effect on the surface using liquid impinging jets for a heated rotating jet. Additionally, effects of jet temperature, Reynolds number, nozzle diameter disk/spacing ratio, nozzle diameter ratios from disk were observed. It was determined that Nusselt Number was increased with increase in Reynolds number, the jet temperature, and the disk spacing ratios of the nozzle diameter. Jahedi et al. [14] performed the cooling of the hot rotating hollow cylinder with one and two rows of water-impinging jets for evaluation of parameters such as sub-cooled water jets, flow rate, rotation velocity, range of jets, and angular position of jets. The use of a two-row jet configuration was envisaged to have higher surface heat flow, a drop-in temperature, and a reduction in the individual jet's mass flow rate up to 50%. Consequently, it was observed that the rotation speed had a more significant effect at 30 rpm. Altaibi et al. [15] analysed the impingement of jets on heat transfer characteristics of the target rotating for Reynolds number, nozzle geometry, spacing between jet exit and the impinging plate, and the angle of impinging of the jet at the exit of the jet flow regime. It was inferred that the jet diameter, the Reynolds number, and the boundary layer thickness were the parameters that influenced the heat transfer characteristic. Baghel et al. [16] studied the heat transfer effect on a semi-cylindrical curved surface by free surface water jet impingement. The ratio of the outer diameter of the cylindrical surface to the outer diameter of the nozzle tube (D/d), and the Reynolds number were used as parameters for this study. It was shown that increase in Reynolds number increased the Nusselt number.

Kilic et al. [17] investigated the heat transfer of a moving plate with high heat flux by using Al2O3-H_2O nanofluid and impingement jets for different Reynolds numbers, nanofluid volume fraction, nanofluid particle diameter and different plate velocities. When the diameter of nanoparticles was reduced from 40 nm to 10 nm, Nusselt number enlarged by 9.1%. The increase in plate speed and flow in the opposite direction augmented the Nusselt number but when flow and plate speed was in same direction, Nusselt number decreased accordingly. Kilic et al. [18] examined the effect of heat transfer and flow characteristics for different Reynolds numbers, inlet configuration and types of different nanofluid. It was obtained that use of Cu-H_2O nanofluid increased average Nusselt number by 3.6%, 7.6%, and 8.5% with respect to TiO_2-H_2O nanofluid, Al_2O_3-H_2O nanofluid and H_2O. Also, as compared to channel flow, 2-jet flow and 4-jet flow, 1-jet flow amplified average Nusselt number by 91.6%, 29.8%, and 57.1% respectively. Al-Zuhairy et al. [19] studied the thermal effect Al_2O_3 water nanofluid on the surface using the twin circular jet technique. The ratio of the nanoparticle concentration and the diameter of the nozzle height (Z/D) was the main parameter for this experimental study. At 0.25 kg/m3 concentration of nanoparticles, it was observed that the Nusselt number enlarged by more than 200% compared to pure water. Kareem et al. [20] studied the heat transfer characteristics on the surface using CuO nanoparticles by single impinging jet. The different Reynolds number, nanoparticle volume fraction and nozzle-jet distance were used as parameters for this study. It was inferred that the use of nanofluids significantly increased the heat transfer effect and heat transfer decreased when the nozzle jet distance increased.

Amjadian et al. [21] investigated the thermal properties of heat transfer and liquid flow of an aluminum disk subjected to constant heat. For this purpose, different Reynolds numbers and nanoparticle concentration were used as parameters. It was detected that convective heat transfer increased by 45% when the CuO nanoparticle concentration was increased from 0.03 to 0.07%. Lv et al. [22] performed the heat transfer experiment by using different volume fractions of 1%, 2% and 3% of SiO_2-H_2O nanofluids and free single jet impingement. Mainly, heat transfer effects were examined for different volume fractions, Reynolds numbers, nozzle to plate distance and the impact angle. Only 3% nanopowder volume fraction of SiO_2-H_2O nanofluid increased the heat transfer coefficient by 40% compared to distilled water. Kilic et al. [23] numerically studied the enhancement of heat transfer on a high heat flux surface using the pulsed jet method with nanofluids for an industrial application. In the study, heat transfer from a flat copper surface was investigated for nanofluids for different particle diameter (Dp=10nm, 20nm, 40nm, 80nm); different Reynolds numbers (Re=12000, 14000, 16000, 18000); different nanofluid types (CuO-water, NiO-water, Cu-water, pure water) and different volume fraction (φ=2%, 4%, 6%, 8%). According to the results; while the Re number increased from 12000 to 18000, the average Nusselt number increased by 28%. Decreasing the particle diameter from 80 nm to 10 nm resulted in a 13.20% increase in the average Nusselt number. It was determined that an increase in nanoparticle volume concentration above 4% did not cause a noteworthy increase in heat transfer.

Shafiq et al. [24] analysed the heat transfer effect of high heat flux of the surface by the impingement jet method. Different Reynolds numbers, different particle diameters of nanofluids and different volume fractions were used as parameters for this study. It was concluded that with the increase in the Reynolds number from 12000 to 18000, there was a 28% increase in the Nusselt number and when volume fraction increased from 2% to 8%, the Nusselt number increased by 7.1%. Moreover, as compared to pure water, Cu-H_2O nanofluid increased the Nusselt number by 8.3%.

Fu et al. [25] studied the temperature drop for different heat flux values during quenching of large-section ultra-heavy steel plate. It has been determined that the distribution of the jet heat transfer zone of the two surfaces in the steel plate is different. It was also investigated that the upper surface adopts jet impingement heat transfer. In order to achieve symmetric cooling, flow rate of the water of the lower surface increased and conduction heat transfer coefficient increased accordingly.

Habio Ma [26] investigated heat transfer mechanism in secondary cooling of continuous casting of steel slab. In his study, the profile of the heat transfer region aligns with the spray pattern, suggesting the dominant role of spray in determining the heat transfer on the slab surface. Outside the spray pattern, the slab loses energy through convection and radiation. While inside the spray pattern, additional heat conduction to droplet and droplet boiling further cools down the hot slab. Six candidate parameters were proposed as thresholds to determine the spray cooling region, but the patterns defined by Nu, HTC (heat transfer coefficient), and surface heat flux are more appropriate to represent the spray-affected area. However, since HTC is one of the primary indicators used in the secondary cooling operation and research, the pattern defined by HTC was selected to evaluate the heat transfer intensity and uniformity.

Tom and Kayabasi [29] studied to remove the heat from the overheated microprocessor with a microchannel heat exchanger using ethylene and compare it with the air-cooled system. In the study the flow paths of the air coming from the outside with the help of the fan were observed. Baysal et al. [30] examined the effect of using turbulator in a counter-flow heat exchanger. Friction factor and pressure drop characteristics were studied for empty and different pitch tabulator models for Re number range of 4000-26000.

When the literature is examined, in order to increase the heat transfer on surfaces with high heat flux, impinging jet, spray cooling method, circular jet cooling applications have been studied. In this study, unlike similar applications in the literature, the swirling jet just below the target surface was designed and studied at different Re numbers in accordance with the industrial application of the roller. Main objective of the present study is to focus on numerical analysis of the cooling surface of the industrial roller with swirling jets through spiral channels to improve the heat transfer performance. Targeted surface temperature of the roller is chosen as $T_{surface}$= 24ºC due to the maximum surface temperature of industrial rollers used in pulp production. It is expected that thermal stress will increase over that temperature and to decrease thermal stress, the temperature difference between inlet and outlet of the surface must be minimum. Therefore, in this study, variation of temperature difference through the surface is investigated to obtain the minimum temperature difference. The model is inspected for different parameters such as different Reynolds number, different heat flux on the surface, and different inlet temperature of coolant.

2. Numerical Model and Mathematical Formulation

Present study is mainly focused on numerical analysis of the cooling surface of the rotating cylinder while providing the fluid flow through spiral channels with swirling jets to increase the heat transfer performance. In this study, target temperature of the roller surface is chosen as 24 ºC for industrial application. This temperature value is chosen to prevent thermal stress on roller surface and damage of the product with high temperature. In this model, the total length of this cylinder is predicted as 410 mm and the outer diameter as 130 mm. 6061 aluminum alloy is chosen as the material of roller. Inlet diameter of coolant is estimated as 29 mm and the diameter of the spiral channels as 10 mm.

In this numerical study, the laminar model of the ANSYS Fluent Computational Fluid Dynamics package program is used. The geometry of the numerically studied model is shown in Figure1.

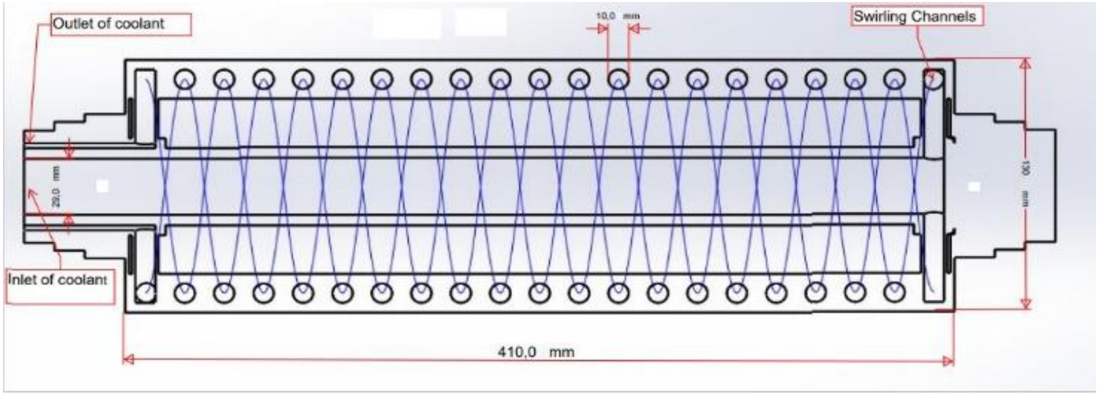


Figure 1. Geometry of the roller

After mesh independence, 843540 nodes and 4054168 elements are obtained. The skewness is closer to 0 and is mostly at rate of 0.142. The orthogonal quality is closer to 1 and mostly at rate of 0.854. The roller surface mesh structure and the water channel mesh structure are shown in Figure 2, Figure 3, Figure 4 and mash independence of average value of the surface for different mash number is shown in Table 1.

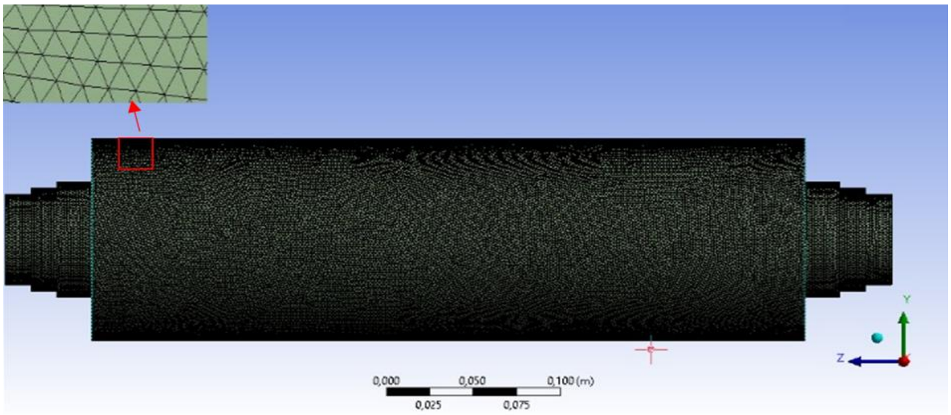


Figure 2. Roller mesh structure

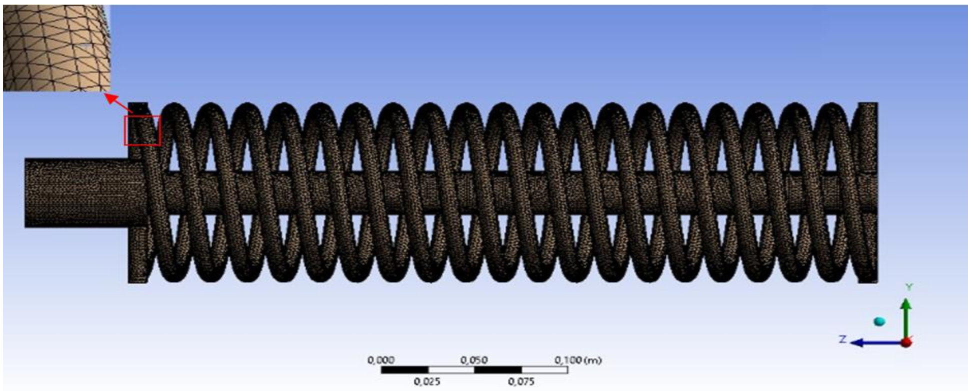


Figure 3. Swirling jets mesh structure

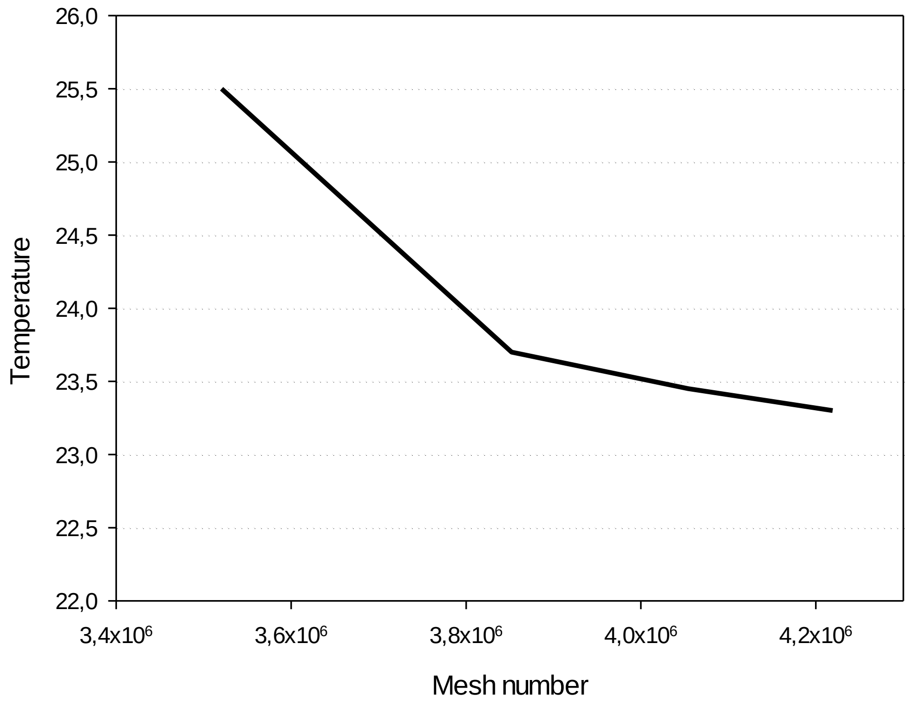


Figure 4. Change of average temperature for different mesh number

Table 1. Mesh independence for different mesh number

Mesh number	$T_{average}$ on roller surface	Orthogonal quality	Skewness
3520500	25.5	0.851	0.146
3852010	23.7	0.853	0.144
4054168	23.47	0.854	0.142
4219278	23.33	0.854	0.142

Following mass balance, momentum balance and energy balance equations [27, 28] are solved in ANSYS fluent while assuming single phase, Newtonian and incompressible fluid along with steady state flow.

Continuity Equation:

$$\frac{\partial \rho}{\partial t} + div(\rho u) = 0 \tag{1}$$

Momentum Equations:

$$\frac{\partial (\rho u)}{\partial t} + div(\rho u u) = -\frac{\partial p}{\partial x} + div(\mu grad u) + S_{Mx} \tag{2}$$

$$\frac{\partial (\rho v)}{\partial t} + div(\rho v u) = -\frac{\partial p}{\partial y} + div(\mu grad v) + S_{My} \tag{3}$$

$$\frac{\partial (\rho w)}{\partial t} + div(\rho w u) = -\frac{\partial p}{\partial z} + div(\mu grad w) + S_{Mz} \tag{4}$$

Energy Equation:

$$\frac{\partial(\rho i)}{\partial t} + div(\rho i u) = -p div u + div(k grad T) + \Phi + S_i \quad (5)$$

Here S_i and Φ denote source term and viscous dissipation term respectively. Viscous dissipation can be calculated as:

$$\Phi = \mu \left\{ \begin{array}{l} 2\left[\left(\frac{\partial u}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial y}\right)^2 + \left(\frac{\partial w}{\partial z}\right)^2\right] + \\ \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)^2 + \left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right)^2 \end{array} \right\} + \lambda(\text{div } \mathbf{u})^2 \quad (6)$$

Turbulent kinetic energy and heat dissipation rate are calculated as follows:

$$\frac{\partial(\rho k)}{\partial t} + div(\rho k \boldsymbol{U}) = div\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right) grad\ k\right] + P_k - \beta^* \rho k \omega \quad (7)$$

$$\frac{\partial(\rho \omega)}{\partial t} + div(\rho \omega \boldsymbol{U}) = div\left[\left(\mu + \frac{\mu_t}{\sigma_\omega}\right) grad\ \omega\right] + \gamma 1\left(2\rho Sij \cdot Sij - \frac{2}{3}\rho\omega \frac{\partial Ui}{\partial xj}\delta_{ij}\right) - \beta_1 \rho \omega^2 \quad (8)$$

3. Results and Discussions

In this section, numerical results are presented based on variation in Reynolds number, surface heat flux and inlet temperature of the fluid.

3.1 Effect of Reynolds Number on Heat Transfer

Different Reynolds numbers (Re=1000, 1300, 1500, 1700) are examined numerically along with heat flux (q''=10000 W/m^2) and inlet temperature of the water (T_{inlet}=5°C). Temperature variation on the surface of the roller for different Re numbers is shown in Figure 5. It is observed that when the Reynolds number is increased from 1000 to 1700, the surface temperature of the roller decreases, and also the temperature distribution on the surface becomes more homogeneous (difference between inlet and outlet temperature of the roller is decreasing).

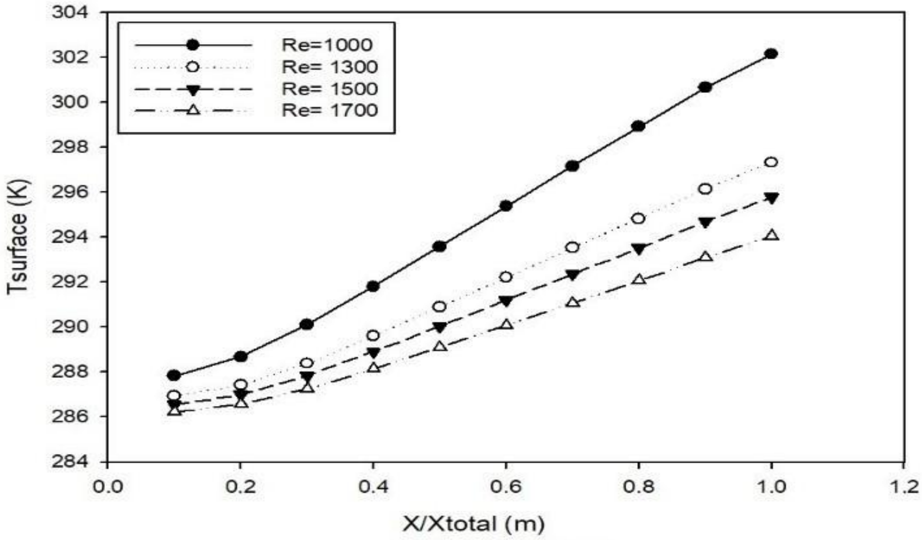
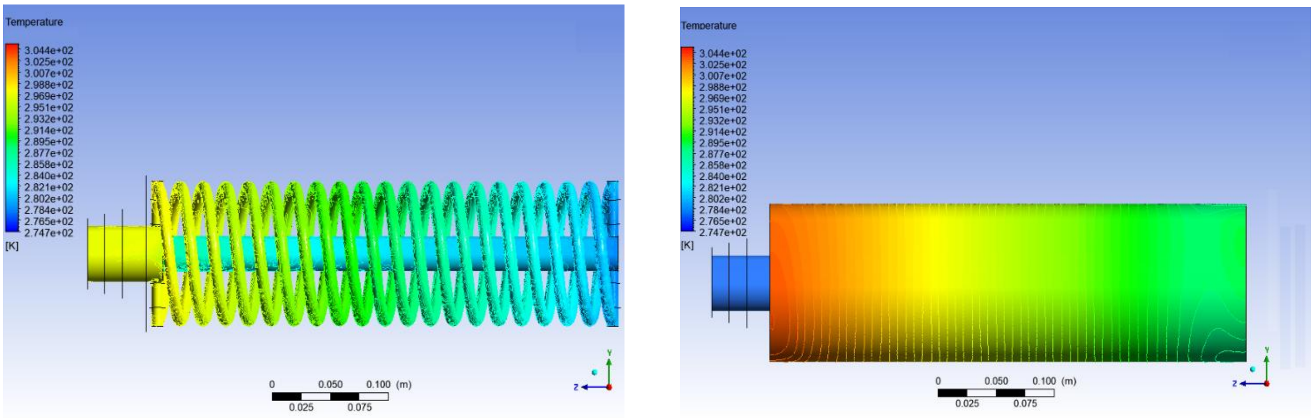


Figure 5. Temperature on the surface of the roller for different Re number

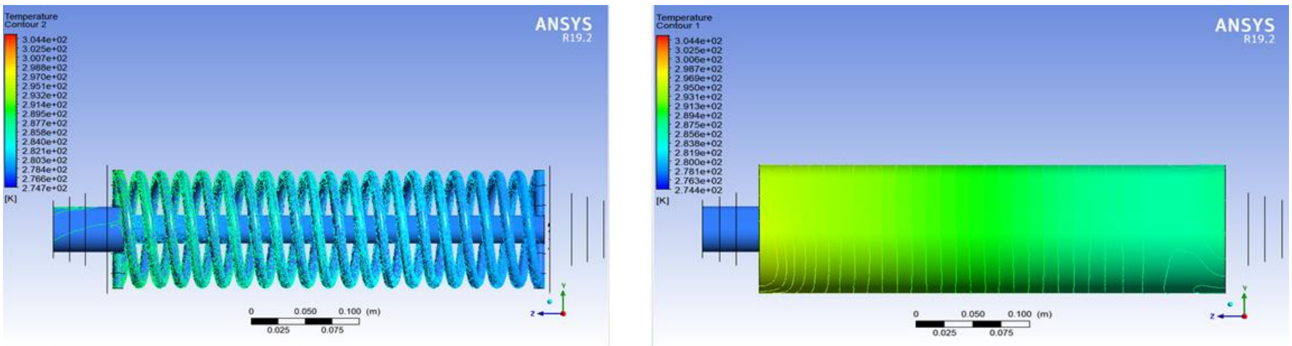
Increasing Re number from 1000 to 1700 causes a decrease of 45.4% on temperature difference, which causes thermal stress, from inlet to the outlet region of the roller surface. Re=1700 shows the best cooling performance but according to the target temperature of the roller surface, Re=1300 is the best solution for the application to keep surface temperature under maximum temperature (24ºC).

Temperature contours for water and roller at Re=1000 and 1700 are presented in Figure 6 and Figure 7 respectively.



(a) (b)

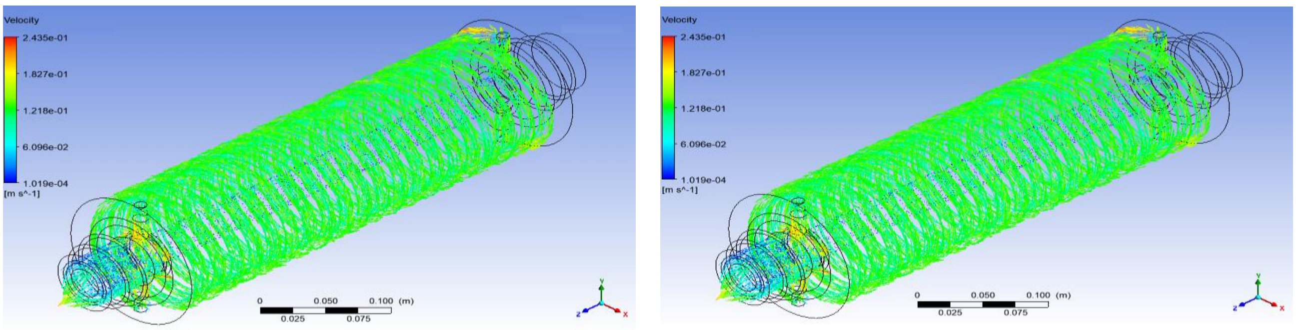
Figure 6. Temperature contours at Re=1000 for (a) swirling jets (b) roller surface



(a) (b)

Figure 7. Temperature contours at Re=1700 for (a) swirling jets (b) roller surface

Velocity vectors are shown for Re=1000 and Re=1700 in Figure 8.



(a) (b)

Figure 8. Velocity vectors at (a) Re=1000 (b) Re=1700

3.2 Effect of Different Surface Heat Flux on Heat Transfer

Different surface heat fluxes (q"=5000, 7500, 10000, 12500 W/m^2) are investigated numerically for Re=1500 and T_{inlet}=5°C. It is obtained that increasing surface heat flux for q"=5000-12500 W/m^2 causes an increase of 149.4% on temperature difference, which causes thermal stress, from inlet to the outlet region of the roller surface. But according to the target surface temperature of roller, cooling parameters of this study can cool the surface of the roller up to the heat flux of q"=12500 W/m^2 effectively. Since velocity of the fluid does not change for this parameter, Temperature contours are only presented. Temperature variation on the surface of the roller for different surface heat flux is shown in Figure 9. Temperature contours for different surface heat fluxes on swirling jets and roller surface are shown in Figure 10 and Figure 11.

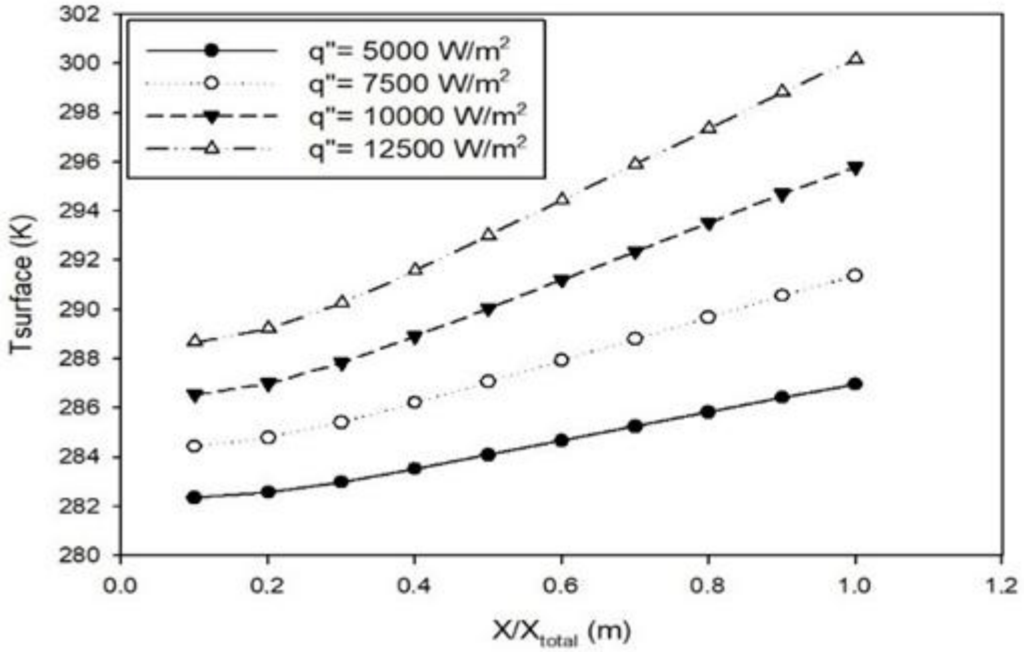
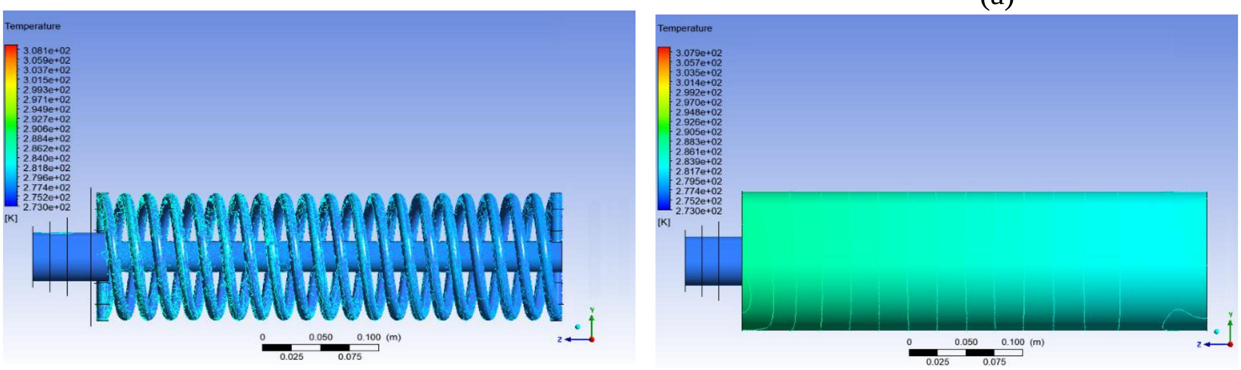


Figure 9. Temperature on the surface of the roller for different surface heat flux values

(a)



(b)

Figure 10. Temperature contours for q"=5000 W/m2 for (a) swirling jets (b) roller surface

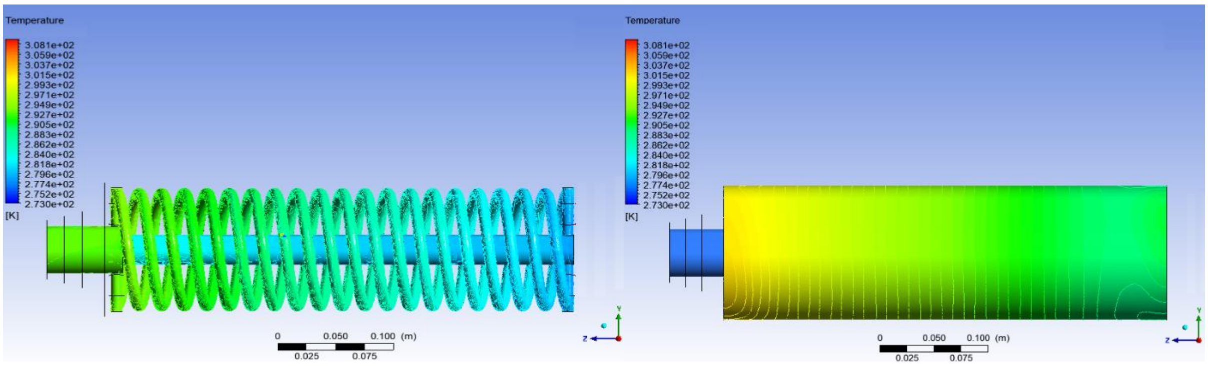


Figure 11. Temperature contours for q"=12500 W/m^2 for (a) swirling jets (b) roller surface

3.3 Effect of Inlet Temperature of fluid on Heat Transfer

Different inlet temperatures of water (T_{inlet}= 5, 7, 10, 15, 20°C) are investigated numerically at Re=1500 and q"= 10000 W/m^2. When the inlet temperature of water increases from 5°C to 20°C, the surface temperature of roller also increases. But temperature difference between inlet and outlet region of the roller surface does not show any significant increase. Besides, temperature difference also doesn't change. So T_{inlet} has not a significant effect of heat transfer. According to the target temperature, for industrial application of this study, it is aimed that maximum surface temperature of roller would be under 27 °C for higher fluid inlet temperature to decrease cooling needs which is chosen in this study. So the best performance for inlet temperature of the fluid is T_{inlet}=10 ºC. Since velocity of the fluid does not change for this parameter, Temperature contours are only presented. Temperature variation on the surface of the roller for different inlet temperature of the fluid is shown in Figure 12.

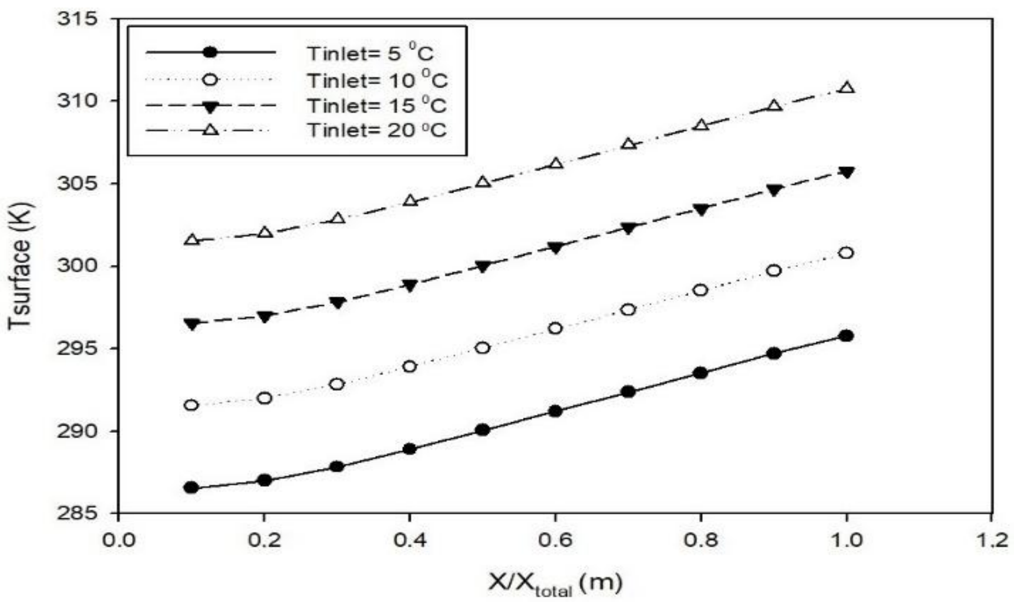
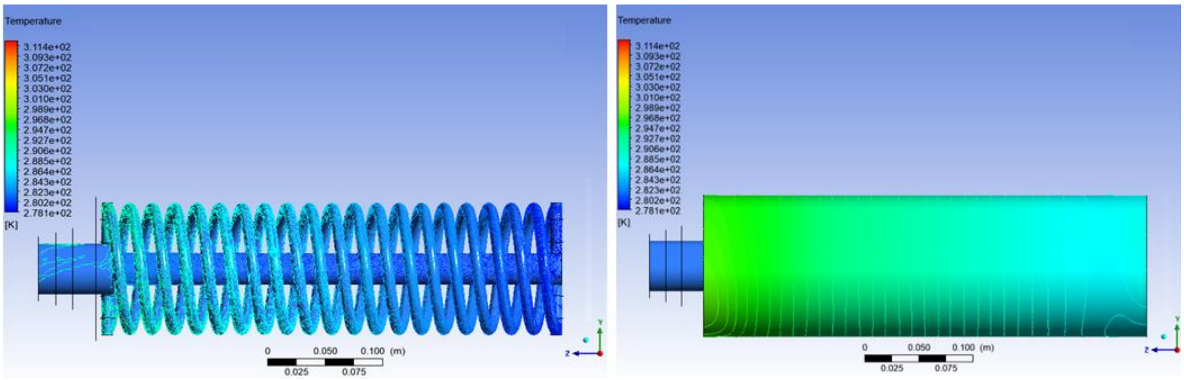


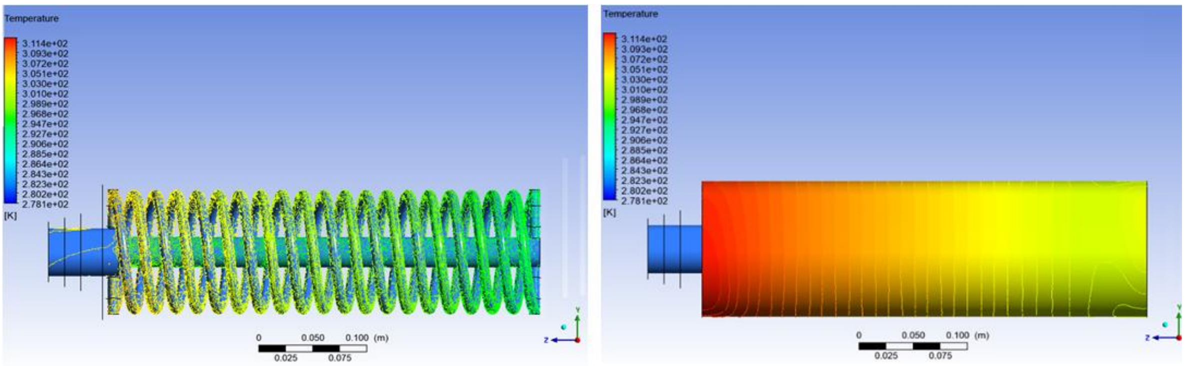
Figure 12. Temperature on roller surface for different inlet temperatures

Temperature contours for different inlet temperature of water at 5°C and 20°C are presented in Figure 13 and Figure 14.



(a) (b)

Figure 13. Temperature contours at T_{inlet}=5°C for (a) swirling jets (b) roller surface



(a) (b)

Figure 14. Temperature contours at T_{inlet}=20°C for (a) swirling jets (b) roller surface

4. Conclusion

This study is mainly focused on numerical analysis of the cooling surface of the industrial roller with swirling jets through spiral channels to increase the heat transfer performance. Numerical model has been investigated for different parameters such as different Reynolds number, different heat flux on the surface, and different inlet temperature of coolant and results are as follows;

(a) Increasing Re number (Re=1000-1700) causes a decrease of 45.4% on temperature deference, which causes thermal stress from inlet to the outlet region of the roller surface. Re=1700 shows the best cooling performance but according to the target temperature of the roller surface Re=1300 is the best solution for the application.

(b) Increasing surface heat flux for q''=5000-12500 W/m2 causes an increase of 149.4% on temperature deference from inlet to the outlet region of the roller surface. But according to the target surface temperature of roller, cooling parameters of this study can cool the surface of the roller for q''=12500 W/m2 effectively.

(c) When the inlet temperature of water increases from 5°C to 20 °C, the surface temperature of roller also increases. But temperature difference between inlet and outlet region of the roller surface does not show any significant increase. Hence, T_{inlet} has not significant effect on heat transfer. According to the target temperature for industrial application which was chosen in this study, the best and optimum inlet temperature of the fluid is 10ºC.

It is evaluated that the results of this study will contribute to design more effective cooled industrial roller. For future studies on cooling of industrial roller, effect of different geometry of cooling channel, effect of different nanofluid as cooling fluid, and effect of rolling velocity of the roller on heat transfer can be investigated.

Authors' Contributions

Kilic, M, Sahin M, and Kilinc, Z., designed the structure. Kilic, M., Ullah, A. and Demircan T. inspected and evaluated the associated laws and data. Kilic, M, Sahin M, Iqbal, M. and Kilinc, Z. created and run the model and solved problems. All authors help to evalute the results and write the publication. All authors read and approved the final manuscript.

Conflict of Interests

The authors declare that they have no Conflict of Interests.

References

[1]. Liu, Y., et al., "Numerical Simulation of Heat Transfer from Rotary Cylinder by Spray Cooling", Mater Sci, 2017, 23(4): 411-415. doi:10.5755/j01.ms.23.4.17322.
[2]. Lu, Q., et al., "Numerical study of a rotating liquid jet impingement cooling system", International Journal of Heat and Mass Transfer, 2020, 163: 120446.
[3]. Luo, J., et al., "Hydrodynamics and heat transfer of multiple droplets successively impacting on cylindrical surface", International Journal of Heat and Mass Transfer, 2021, 180: 121749.
[4]. Du, C., et al., "Numerical study on vortex cooling flow and heat transfer behavior under rotating conditions", International Journal of Heat and Mass Transfer, 2017, 105: 638-47.
[5]. Zhao, X., et al., "Experimental investigation of surface temperature non-uniformity in spray cooling", International Journal of Heat and Mass Transfer, 2020, 146: 118119.
[6]. Modak, M., et al., "An Experimental Investigation on Heat Transfer Characteristics of Hot Surface by Using CuO–Water Nanofluids in Circular Jet Impingement Cooling", Journal of Heat Transfer 2017, 140(1). doi:10.1115/1.4037396.
[7]. Selimefendigil, F., Öztop, H., "Analysis and predictive modeling of nanofluid-jet impingement cooling of an isothermal surface under the influence of a rotating cylinder", International Journal of Heat and Mass Transfer, 2018, 121: 233-245.
[8]. Mahdavi, M., et al., "Thermal analysis of a nanofluid free jet impingement on a rotating disk using volume of fluid in combination with discrete modelling", International Journal of Thermal Sciences, 2020, 158: 106532.
[9]. Baghel, K., et al., "Inclined free-surface liquid jet impingement on semi-cylindrical convex curved and flat surfaces: Heat transfer characteristics", International Communications in Heat and Mass Transfer, 2021, 121: 105116.
[10]. Pachpute, S., Premachandran, B., "Effect of number of round jets on impingement Heat Transfer from a Heated Cylinder", Applied Thermal Engineering, 2019, 162: 114308.
[11]. Jahedi, M, Moshfegh, B. "Experimental study of quenching process on a rotating hollow cylinder by one row of impinging jets", 9th World Conference on Experimental Heat Transfer, Fluid Mechanics and Thermodynamics, 12-15 June, Iguazu Falls, Brazil, (2017).
[12]. Patil, V. S., Vedula, R. P., "Local heat transfer for jet impingement onto a concave surface including injection nozzle length to diameter and curvature ratio effects", Experimental Thermal and Fluid Science, 2018, 92: 375-89.
[13]. Jiang, L., et al., "Numerical investigation of conjugate heat transfer on a rotating disk under round liquid jet impingement", International Journal of Thermal Sciences 2021, 170: 720-734.

[14]. Jahedi, M., Moshfegh, B., “Quenching a rotary hollow cylinder by multiple configurations of water-impinging jets”, International Journal of Heat and Mass Transfer, 2019, 137: 124-37.
[15]. Altaibi, H.M., et al. “Review paper on heat transfer and flow dynamics in subsonic circular jets impinging on rotating disk”, Energy Reports, 2020, 6: 834-842.
[16]. Baghel, K., “Heat transfer characteristics of free surface water jet impingement on a curved surface”, International Journal of Heat and Mass Transfer, 2020, 164: 120487.
[17]. Kilic, M, Ali, H., “Numerical investigation of combined effect of nanofluids and multiple impinging jets on heat transfer”, Thermal Science, 2019, 23: 3165-73.
[18]. Kilic, M, Abdulvahitoglu, A. “Numerical investigation of heat transfer at a rectangular channel with combined effect of nanofluids and swirling jets in a vehicle radiator”, Thermal Science, 2019, 23: 3627-37.
[19]. Al-Zuhairy, R. C., et al., “Al2O3-water nanofluid heat transfer enhancement of a twin impingement jet”, Case Studies in Thermal Engineering, 2020, 19: 100626.
[20]. Kareem, Z.S., et al., “Heat transfer enhancement in single circular impingement jet by CuO-water nanofluid”, Case Studies in Thermal Engineering, 2019, 15: 100508.
[21]. Amjadian, M., et al., “Heat transfer characteristics of impinging jet on a hot surface with constant heat flux using Cu2O–water nanofluid: An experimental study”, International Communications in Heat and Mass Transfer 2020, 112: 104509.
[22]. Lv, J., Hu, C., Bai, M., Zeng, K., Chang, S., Gao, D., “Experimental investigation of free single jet impingement using SiO2-water nanofluid”, Experimental Thermal and Fluid Science, 2017, 84: 39-46.
[23]. Kilic, M., Özcan, O., “Numerical study of the joint effect of nanoplastic and multiplier jets for different parameters”, Journal of The Faculty of Engineering and Architecture of Gazi University, 2019, 34: 1501-1516.
[24]. Shafiq, F., et al., “Natural Convection Heat Transfer in an Enclosed Assembly of Thin Vertical Cylinders – A CFD Study”, Chemical Engineering and Technology, 2020, 43: 1648-1658.
[25]. FU, Tianliang, et al. “Experimental study on temperature drop during roller quenching process of large-section ultra-heavy steel plate”, Science Progress, 2021, 104(2): 00368504211009330.
[26]. Ma, H., “On heat transfer mechanisms in secondary cooling of continuous casting of steel slab”, School of Mechanical Engineering West Lafayette, Indiana 2021.
[27]. Kilic, M., Ullah, A., “Numerical investigation of effect of different parameter on heat transfer for a crossflow heat exchanger by using nanofluids”, Journal of Thermal Engineering 2021, 7: 1980-1989.
[28]. Shafiq, F., Ullah, A., Nadeem, M., Khan, A., “Natural Convection Heat Transfer in an Enclosed Assembly of Thin Vertical Cylinders – A CFD Study”, Chemical Engineering and Technology, 2020, 43(8): 1-12.
[29]. Tom, B., and Kayabaşı, E., “Design and Simulation of Ethylene Cooled Microchannel Heat Exchanger for Micro Processors”, El-Cezeri, 2021, 8(3): 1243-1253.
[30]. Baysal, E., Solmaz, Ö., and Ökten, M., “İç İçe Borulu Isı Değiştiricide Zıt Akışta Türbülatör Kullanımının Basınç Değişimine Etkisinin Sayısal Olarak İncelenmesi” El-Cezeri, 2021, 8(2): 817-826.