Int. J. Nonlinear Anal. Appl. 15 (2024) 5, 297–310
ISSN: 2008-6822 (electronic)
http://dx.doi.org/10.22075/ijnaa.2022.28781.3986

Investigating the distance of the injection jets to the target plate and their number and the use of twisted tape in the rotating jet at various angles and its effect on the average Nusselt number and heat transfer

Hamid Nasirzadeh, Mohammad Eftekhari Yazdi*, Arash Mirabdolah Lavasani

Department of Mechanical Engineering, Central Tehran Branch, Islamic Azad University, Tehran, Iran

(Communicated by Seyed Hossein Siadati)

Abstract

There are various industries, each of them is concerned with heat transfer, and in many cases, their goal is to increase the rate of heat transfer. In this study, one, two or four injection jets were used together for cooling and the results were compared in terms of the average Nusselt number. In the following, by placing the twisted tape at different angles of 180°, 360° and 720° in the injection jet, the average Nusselt number and the minimum and maximum temperature were checked. The results showed the diameter of the fluid outflow from the nozzle increases by moving away from the jet opening and approaching the hot plate due to the pressure distribution. When we using two jets in the vicinity, the Outputs from each vortex is created in this place. When using four jets, the outflow from the jet hits the hot plate and creates vortices. With increase in the number of jets, the Nusselt number and the maximum temperature on the hot surface also increase. The results showed that the average Nusselt number with the angle of the twisted strip that is placed inside the rotating jet tube increases at 180° and decreases at 720° and 360°. In all cases using twisted strips, the maximum and minimum temperatures of the hot plate are higher than when the non-rotating jet is injected on the hot plate. As the jet-to-plate spacing plate increases, the Nusselt number decreases.

Keywords: heat transfer, Nusselt number, injection jet, rotating jet
2020 MSC: 80Axx

1 Introduction

Increasing the rate of heat transfer is important in various industries that are especially Involved with the problem of heat transfer. In many cases, jets are used that are injected on a plate and thus the heat transfer rate is changed. In addition to simple jets injected with different sections, jets can be considered as twisting, so that the fluid is rotated around the axis passing through the center of the jet before exiting the jet outlet nozzle. Therefore, changes in the heat transfer rate have been created that can be used by different industries. The outflow from the jet has different applications in different industries. Sometimes the energy output from the jet is used to move the plane and launch

*Corresponding author
Email addresses: hamid.mechanic.67@gmail.com (Hamid Nasirzadeh), moh.eftekhari_yazdi@iauctb.ac.ir (Mohammad Eftekhari Yazdi), arashlavasani@iauctb.ac.ir (Arash Mirabdolah Lavasani)

Received: October 2022 *Accepted:* December 2022

rockets. In some cases, the exit jet is used to spread the pollutant, like what is seen in different power plants. In some cases, the jet used to cool the surfaces in such a way that by injecting the jet on a hot plate, it causes changes in the hydraulic and thermal boundary layer formed on the surface and heat transfer increases. Figure 1 shows how a jet is injected on a hot surface.

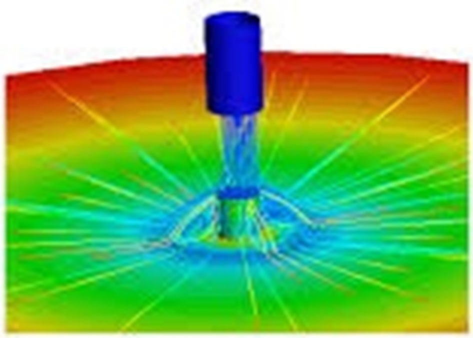

Figure 1: Exiting jet from the nozzle that is injected on the plate

During jet injection on a plate, different regions of the flow field can be divided into different regions. As the flow exits the nozzle, the potential core is first formed, where the flow exiting from the less jet is less affected by the surrounding environment. As the jet penetrates the fluid, the outgoing jet is affected by the surrounding fluid. This region is known as free jet region because the wall has not yet affected the jet and Whatever that is happens is the result of the reaction of the jet to the surrounding fluid. In the free zone, it can be seen that the Cross-sectional area of the jet undergoes changes and becomes wider. As the jet approaches the wall, an area known as the affected area is created. Here the jet go near the wall and its movement is influenced by the wall. Therefore, the cross-sectional area grows faster and the jet exits deflected. At the point where the jet hits the wall, there is a stagnation point where the fluid coming out of the nozzle stops and its speeding becomes zero. The wall causes the direction of the jet to change and a wall jet is created that moves away from the center of the injected jet along the wall.

Jet injection topic causes the temperature boundary layer formed on the surface to lose its shape and become unstable. Also, with the local increase of the fluid velocity in the vicinity of the surface, the displacement heat transfer coefficient increases and therefore the heat transfer rate increases. The jet can be injected on the hot surface in different ways. One of the types of jet is the twisting jet, in which the fluid rotates before exiting the nozzle during a twisting movement and then leaves the nozzle. It is shown in fig 2 how to help create the rotational movement of the jet by digging the grooves with different angles. In this figure, a specific example of the inner wall of the twisting jet and different shapes are considered for this purpose.

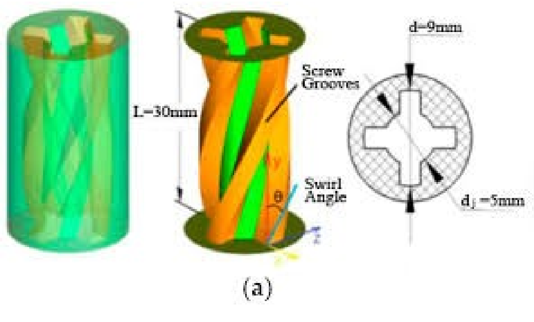

Figure 2: The inner wall of the nozzle to create twisting motion

The twisted jet can cause changes in the physics of the flow, which causes changes in the heat transfer rate. The flow field and temperature resulting from jet injection on the hot plate can be strongly affected by the number of grooves, the depth of the grooves, and the angle of the grooves. Also, the Reynolds number that the fluid exits from the jet is very effective, and at different twisting angles of the jet, the fluid with different velocity exiting the nozzle has different performance.

2 Literature review

Jambunathan et al. [5] conducted a review study regarding the issue of jet injection on a hot plate. They investigated the jet with different Reynolds in the range of 5000 to 124000 in such a way that the distance of the hot plate to the nozzle was changed in the range of 1.2 to 16 times the diameter of the nozzle. They showed the Nusselt number increases as the Reynolds number increases in all cases. In another study by Martin [11], the cooling of a plate by injection jets was investigated. He stated that two different types of jets are used for cooling a plate, which are plate jets and nozzles with different cross-sections. The jets are placed in a different arrangement with respect to the screen and are injected on it. Heat transfer from the plate depends on different parameters. Such as the speed, the type of fluid that comes out of the jet, or the geometric shape and configuration of the jet that is injected on the plate. Lightell and Webb [11] investigated the flow and heat transfer resulting from jet injection on a flat plate in such a way that the distance between the plate and the nozzle was less than the diameter of the nozzle outlet. In the study, it is shown that the Nusselt number changes as a function of Reynolds number. They obtained the Nusselt number locally and at different radial distances from the point of impact of the jet to the plane. Lee et al [9] investigated the effects of nozzle diameter on fluid flow and heat transfer in such a way that the jet exiting from a nozzle with a circular section is injected on a plate with a constant flux. They measured the temperature of the hot surface experimentally. Reynolds number of the fluid exiting from the nozzle was considered equal to 23000 and a uniform flux was applied on the hot plate. The diameter of the nozzle varied between 1.36 and 3.40 cm, and the distance between the nozzle and the screen varied between 2 and 14 times the diameter of the nozzle opening. They showed that increasing the diameter of the nozzle increases the rate of heat transfer in the stagnation region. Goldstein and Franchett [3] In an experimental study,addressed the issue of heat transfer and fluid flow near a flat plate in the condition that a fixed heat flux was applied to the plate and even injected on it. They considered the jet at different angles and with the numerous technique report the numerical value of the temperature on the surface. They showed that the point on the plate with the maximum heat transfer is shifted by changing the angle of the jet from the point of contact of the jet with the plate. Koseoglu & Baskaya [6] The flow and heat transfer of the fluid on a plate were investigated in the case that the jet was injected from the nozzle with different sections on the plate. The cross section of the nozzle has different circular, oval and rectangular shapes. 9 different jets are considered from which the fluid is discharged and injected on the plate. The study is carried out in two experimental and numerical modes, and in its numerical method, turbulence equations are used to simulate the phenomenon of turbulence in the flow. Xu et al [18] investigated a two-dimensional model of a strip jet whose geometry is shown in Fig 3.

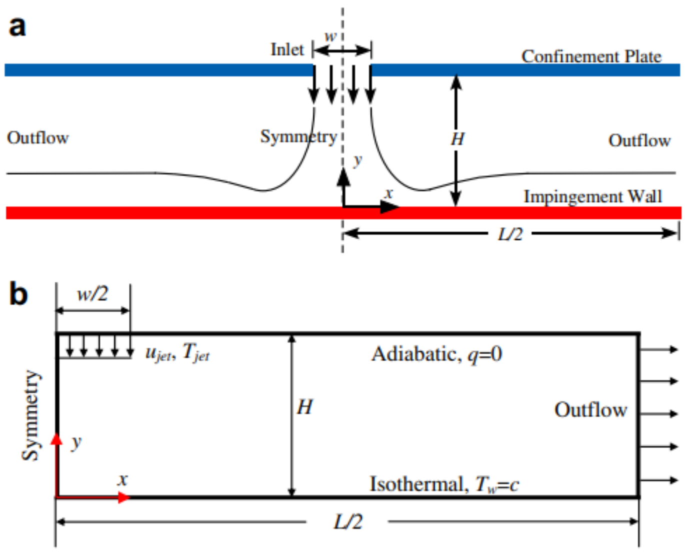

Figure 3: Geometry and computational space related to slot jet injection [14]

The outflow from the jet is considered as a pulse that change with time changes. In this case, the Nusselt number changes in a wide range of Reynolds number, oscillation frequency and the distance from the nozzle to the plate. Weigand and Spring [15], investigated the heat transfer from a plate where a large number of jets are injected together on the plate. The results of this study showed that the Nusselt number fluctuates when moving on the hot plate because at some points on the plate, the jet is directly injected, which leads to an increase in the Nusselt number in the surrounding area, and with the distance from the center of the jet injection, The Nusselt number will decrease.

Molana and Banooni [11] showed that the shape of injected jet on the plate is strongly influenced by the boundary conditions governing it. When the liquid jet is injected on the plate, liquid droplets are formed, which reduces this phenomenon of heat transfer. This is while gas jet injection does not lead to the creation of particles. It is shown in Fig 4 that in an injection jet on a hot plate, the rate of heat transfer increases with the increase of Reynolds number, and it is also seen that the use of nanofluid with a higher concentration leads to an increase in heat transfer.

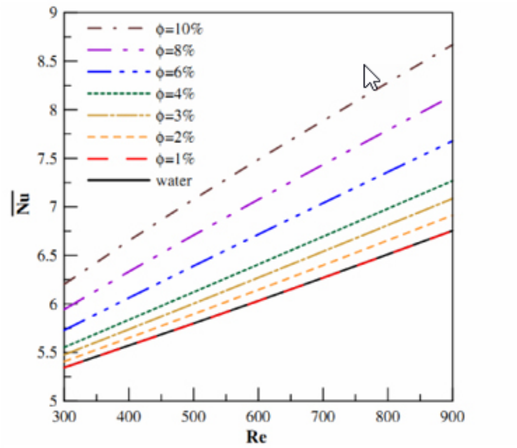

Figure 4: Changes of nusselt number compared to Reynolds number and nanofluid concentration [18]

It is shown in Fig 5 that the increase in Reynolds number and concentration of nanofluid increases the pump power. Therefore, in the problem related to the increase of Reynolds number and concentration of nanofluid, two parameters of increase of Nusselt number and increase of pump power should be investigated at the same time.

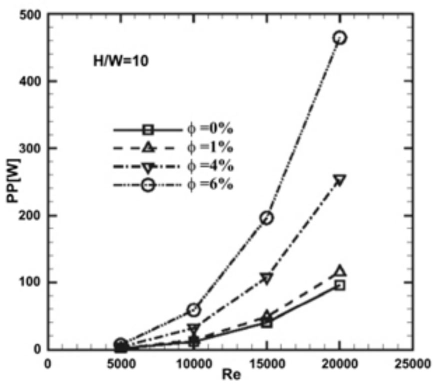

Figure 5: Changes in power required to pump jet fluid with increasing Reynolds number and nanofluid concentration [9]

Nanan et al. [14] investigated the heat transfer from a plate in such a way that a jet is injected on it. They placed two strips in the jet before injection and twisted the strip at different angles in the jet to show how the heat transfer and flow pattern are affected by the twisting of the strips in the jet. The results showed that for smaller distances from the jet to the plate, the effects of twisting on heat transfer are feeling more. Ianiro and Cardone [4] investigated the effects of rotating jet on heat transfer from a plate. They obtained their results for a Reynolds number of 28,000 and evaluated the fluid at different swirls while varying the jet-to-plate distance. Their results showed that the rotating jet reduces the heat transfer in all cases, while the results showed a more uniform heat transfer from the surface due to the rotation of the jet. Le et al. [8] investigated the heat transfer from a plate in a state where a twisting jet is injected on it. They obtained their results in the case where the nanofluid exit jet was with different concentrations. The results showed with the increase in nanofluid concentration, the rate of heat transfer from the plate increases. Barrau et al. [1] investigated the outflow from the jet in such a way that the geometric shape of the jet aperture was considered different. The results of the flow and heat transfer parameters in the channel were investigated in the case where different configurations were considered for the jet outlet.Nakharintr et al. [12] investigated the effects of the distance between the jet and the plate on the heat transfer and the way it hits the plate. They considered a laboratory system as shown in Fig 6.

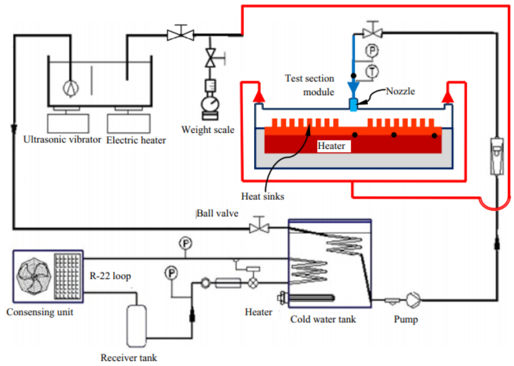

Figure 6: Laboratory device of Nakharintr et al [12]

It is shown in fig 7 that with the increase of the mass flow out of the nozzle, the Nusselt number on the plate increases.

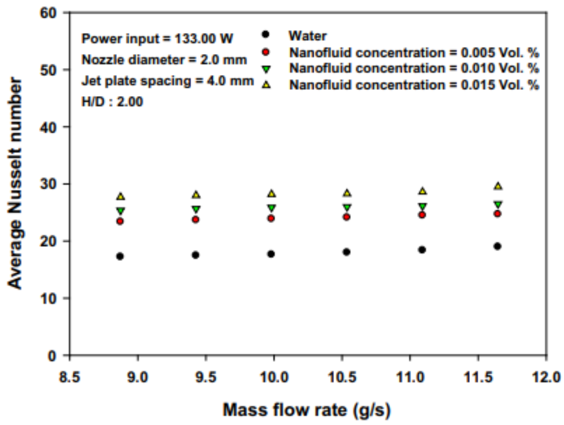

Figure 7: Nusselt number changes in relation to the mass flow of the jet [12]

Fig 8 shows the effects of nanofluid concentration and fluid output from the jet on pressure drop. As can be seen, with the increase in mass flow rate and concentration of nanofluid, we will see that the pressure drop increases.

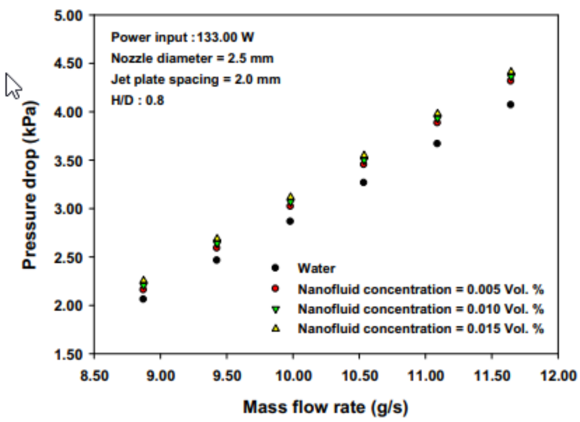

Figure 8: Changes in pressure relative to the mass flow of the jet [12]

According to fig 9, it can be seen that the pressure drop decreases as the distance of the jet from the screen increases.They showed that, in general, the Nusselt number and pressure drop show different behaviors with respect to different parameters.

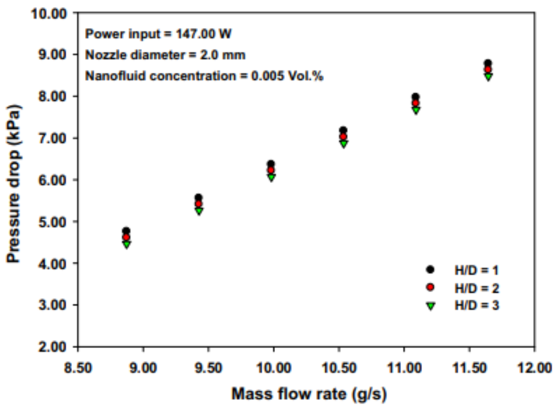

Figure 9: Changes in pressure drop in relation to the mass flow of the jet and the ratio of the distance of the jet from the plate to its diameter [12]

In their study, Le et al. [8] investigated the exit jet from an orifice in a way. This study was focused on two parts, in the first part, the exit jet was investigated at different distances from the exit. They investigated the jet at three different distances as shown in fig 10.

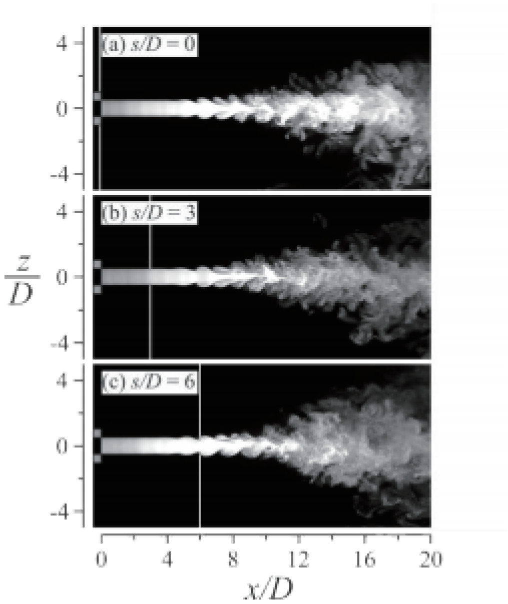

Figure 10: The jet studied by Le et al, with different distances from the jet exit [7]

Fig 11 shows how the velocity of the fluid exiting the jet changes with the distance from the plate. For this purpose, three different axes have been considered for measurement and the fluid velocity has been calculated on these axes. These axes are located at different distances from the symmetry axis of the jet. It can be seen that by moving away from the jet, the velocity of the fluid changes and decreases a lot. Also, can be seen that by moving away from the axis of symmetry of the jet, the velocity of the fluid decreases slightly.

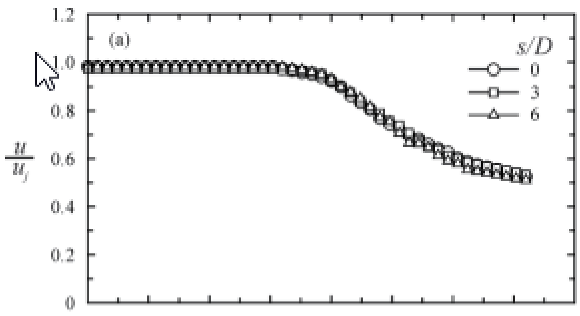

Figure 11: Changes in jet speed at different distances from the axis of symmetry of the jet [18]

Fig 12 shows the temperature contour changes at different distances from the axis of symmetry of the jet. For this purpose, a wall is considered at different distances from the jet axis, on which the fluid exiting from the jet moves. As can be seen, by moving away from the axis of symmetry of the jet, a wider area is affected by the fluid flow and therefore the temperature decreases in a larger area.

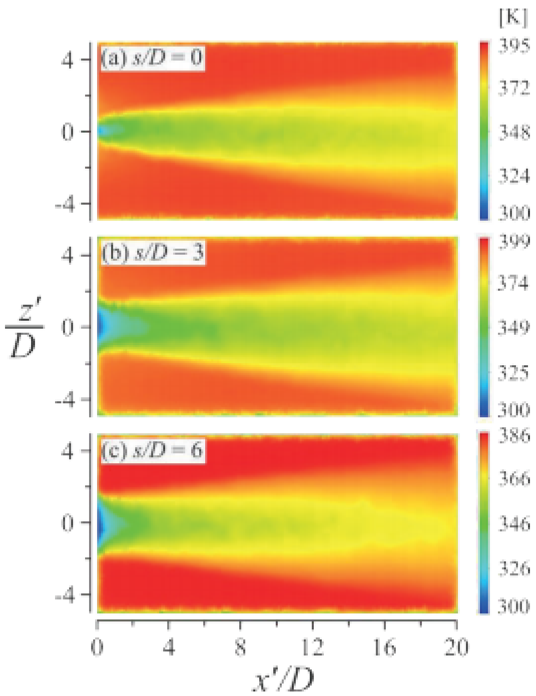

Figure 12: Temperature contour changes at different distances from the jet symmetry axis [18]

Naphon et al [14] built a laboratory device whose purpose was to investigate the heat transfer caused by jet injection on a plate. They considered a jet of different diameter injected on a hot plate fixed at a certain distance from the jet. In their studies, they used nanofluid with different concentration to show how the concentration of nanofluid will affect the heat transfer regime in different geometrical conditions.Fernandez et al. [2] used the complete solution approach of Navier-Stokes equations to clarify the role of surface tension in the formation of circular hydraulic jump. If the surface tension, which depends on flow conditions, fluid properties, and downstream conditions, is above a critical value, no symmetric circular jump will occur.

The innovative aspect of this research is:

- The cooling of a flat plate is done using injection jets on the plate. The flow and temperature field of the fluid exiting the nozzle is in contrast with the hot plate that is in front of the nozzle. After examining a sample of jet injection on the screen, we place grooves on the inner wall of the nozzle, which causes the flow inside the nozzle to rotate under the influence of these grooves. Then, the effect of placing more than one nozzle is studied to show how the flow pattern changes in the solution domain by placing different jets together. Two general ideas are followed in this research, which are stated below.

- Using one, two or 4 injection jets together and comparing the results in terms of the average Nusselt number in the works done by other researchers. As stated in the previous studies, the effects of several jets placed together on heat transfer have not been investigated.

- Placing the twisted strip at different angles of 180°, 360° and 720° in the injection jet in order to check the average Nusselt number and the minimum and maximum temperature.

Research methodology

This research is an applied type and the results of which can be used by engineers in the heat transfer problem. The information used in this article has been extracted from articles that have been published in various Journal.This article examines the flow and heat transfer of the fluid exiting the jet and hitting a hot plate. The research problem is shown as a schematic geometry in Figure 13. As it can be seen, before injecting on the hot plate, the jet passes through a channel and exits through a circular cross section and is injected on the hot plate.

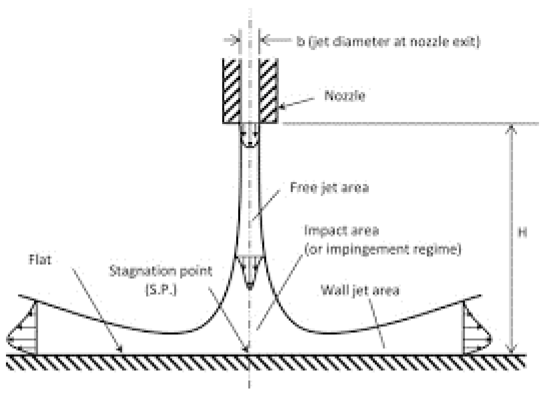

Figure 13: Schematic of injection jet on a hot plate [12]

The main idea in this article is to add grooves at different angles in the inner part of the jet structure. Then, the effect of these rotations on heat transfer and flow field is investigated. Nusselt number as well as flow and heat transfer under the influence of twisting flow are investigated. In the following, two twisting jets are placed together in a special state and a twist angle, and the physics of the problem and existing vortices are investigated. In order to extract and analyze the results, it is necessary to solve the continuity, momentum, energy and turbulence equations simultaneously. For this purpose, it is necessary that the equations are discretized and solved numerically and by repetition. The main equations are given below.

The continuity equation that shows the conservation of mass in the system can be shown as follows:

$$\frac{\partial}{\partial x}(\rho u)+\frac{\partial}{\partial y}(\rho v)+\frac{\partial}{\partial z}(\rho w)=0. \tag{2.1}$$

The momentum equation that represents the movement of the fluid under the influence of the forces acting on it is written in three dimensions, which is given below for the momentum in $\boldsymbol{X}$ direction, $\boldsymbol{Y}$ direction and Z direction respectively.

$$\begin{aligned}
&u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}+\frac{1}{\rho}\frac{\partial P}{\partial x}-v\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)+\frac{\partial}{\partial x}(\rho u'u')+\frac{\partial}{\partial y}(\rho u'v')+\frac{\partial}{\partial z}(\rho u'w')=0\\
&u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}+\frac{1}{\rho}\frac{\partial P}{\partial y}-v\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)+\frac{\partial}{\partial x}(\rho v'u')+\frac{\partial}{\partial y}(\rho v'v')+\frac{\partial}{\partial z}(\rho v'w')=0\\
&u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}+\frac{1}{\rho}\frac{\partial P}{\partial z}-v\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)+\frac{\partial}{\partial x}(\rho w'u')+\frac{\partial}{\partial y}(\rho w'v')+\frac{\partial}{\partial z}(\rho w'w')=0.
\end{aligned} \tag{2.2}$$

Since the main goal in this problem is heat transfer, it is necessary to solve the energy equation, which can be expressed in different forms based on enthalpy and temperature, which differ only in the form of expression and are the same in nature. The energy equation is given below.

$$\frac{\partial}{\partial x}(\rho uh) + \frac{\partial}{\partial y}(\rho vh) + \frac{\partial}{\partial z}(\rho wh) = \frac{\partial}{\partial x}(K\frac{\partial T}{\partial x}) + \frac{\partial}{\partial y}(K\frac{\partial T}{\partial y}) + \frac{\partial}{\partial z}(K\frac{\partial T}{\partial z}). \tag{2.3}$$

Because injecting a jet into another fluid creates vortices, therfor necessary to solve the turbulence equations as well. Here, non-linear $k-\varepsilon$ equations are used to model turbulence.

$$\frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j}\left(\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right) + G_k + G_b - \rho\varepsilon \tag{2.4}$$

$$\frac{\partial}{\partial x_j}(\rho \varepsilon u_j) = \frac{\partial}{\partial x_j}\left(\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right) + \rho C_{1\varepsilon} S_\varepsilon - \rho C_{2\varepsilon}\frac{\varepsilon^2}{k+\sqrt{v\varepsilon}} + C_{1\varepsilon}\frac{\varepsilon}{k}(C_{3\varepsilon}G_b). \tag{2.5}$$

By numerically solving the above equations on the displayed geometry, the flow and heat fields can be obtained. For this purpose, Gambit and Fluent software are used. The geometry is drawn in Gambit and then the discretized equations are solved by the second order discretization method on the computing nodes by repeating method.

Research question

Does the rotating jet increase the heat transfer rate and in which of the angles of the grooves, the most suitable heat transfer rate occurs?

What is the effect of changing the angle of the grooves on the jet wall, which leads to the formation of a rotating jet, on the rate of heat transfer and the temperature and flow field?

What is the effect of placing jets next to each other on heat transfer and flow pattern?

To check the flow and heat transfer of the fluid exiting from the jet, depending on the flow regime, the type of exit fluid and side flow, the equations used for modeling are different. Also, according to which flow parameters are important for calculation, special equations should be used. If we do the calculations for a jet with a Reynolds number lower than the laminar flow, the Navirastox equation can be used in the laminar state, but in general, turbulence models should be used to solve it. These equations have been used in various studies [16].

When checking the jet, the temperature difference between the fluid coming out of the jet and the surface it collides with may be decisive. Therefore, here and for all jets with these conditions, the energy equation should also solved. When the jet is injected into a flow, it can be assumed that around the jet exit, the momentum of the gas plays the main role in the movement, and as we move away from the origin, the contribution of the buoyancy force increases in the movement and determines the path of the gas. Normally, in the analysis of the gas exiting from the jet, there should be a continuity equation, three Navirastox equations, an energy equation, for each type of exit gas different from the gas in which the fluid is injected, an infiltration or infiltration equation and in some cases an equation Solved discretely. Of course, in this article, the nature of the outlet gas and the fluid in which the jet is discharged is the same, and therefore there is no need to solve multiphase equations.

Choosing the turbulence solution method for the first numerical simulation is to use the Navier-Stokes equations. This modeling has been used in this research due to its simplicity and at the same time being practical. The average product of the fluctuating components of the velocity creates the Reynolds Stress. To calculate the Reynolds Stresses, one of the turbulence models should be applied.

Results

As shown in figure 14, the jet is injected into a chamber, which is a rectangular cube with dimensions of 140 mm x 140 mm, which has a height of 7 mm. After the jet travels a distance under the influence of the grooves, it takes a rotational movement and is injected into the chamber between two plates through an opening and is injected onto a plate with dimensions of 140 mm x 140 mm. The plate on which the injection takes place has a higher temperature than the fluid exiting the jet, and the exit fluid cools the plate.

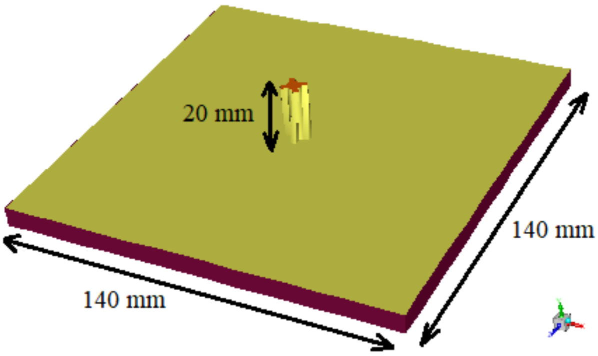

Figure 14: General dimensions of the computing space (researcher's findings)

In order for the jet to take a rotational movement, it is necessary to have a certain amount of rotational movement in a tube, which is done with the help of the grooves inside the tube. After the fluid acquires its rotational motion, it exits the tube and is injected on the plate. Rotational movement occurs inside a 20 mm long channel. The grooves placed inside the tube are such that the inner radius of the groove is equal to 5 mm, while the radius of the outer part is equal to 9 mm. Fig 15 shows how the grooves are placed inside the tube.

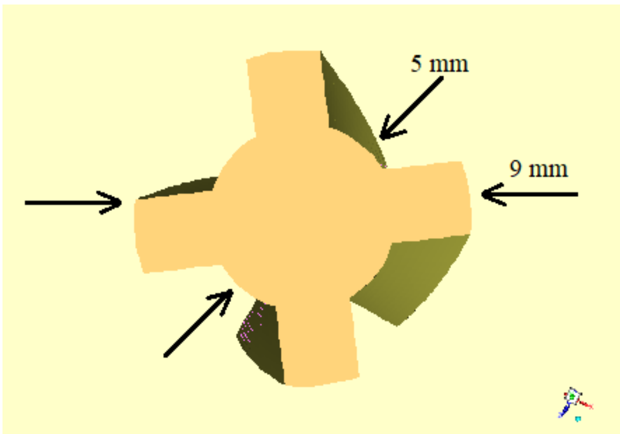

Figure 15: View of the grooves placed in the inner part of the pipe (researcher's findings)

In this research, unorganized networks have been used to network the computing space. The highest concentration is where the gradient of parameters such as speed, pressure and temperature is higher.

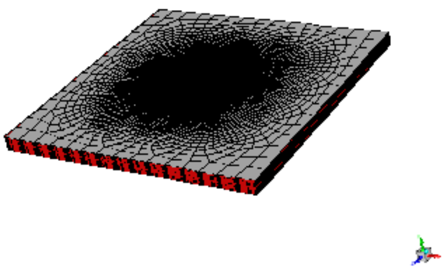

Figure 16: A general view of the grid of solution space (researcher's findings)

In order to obtain complete information of the grid (space and geometry), one grid was drawn around the exit jet

(Fig 17). Near the place where the jet meets the screen, the computing nodes should be closer to each other, and the distance from the meeting point of the jet to the screen increases, the distance between the computing nodes increases.

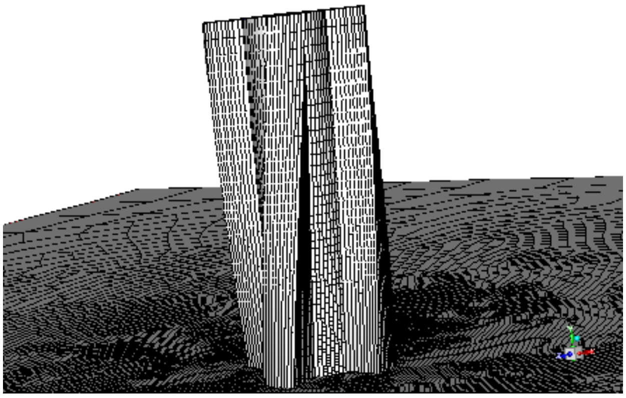

Figure 17: A view of the grid near the jet injection place (researcher's findings)

In order to obtain information from the computational nodes inside the jet, the inlet part of the jet is meshed according to the method of Fig 18. This meshing is extended along the jet to cover its entire volume.

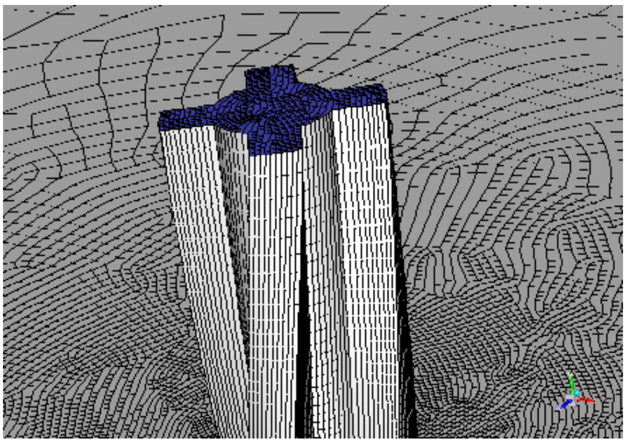

Figure 18: A view of the grid on the upper surface of the jet (researcher's findings)

The boundary conditions of the computational space have been such that the air fluid enters the jet channel with a length of 20 mm at a speed of 16.9 m/s and flows in it. Grooves are placed on the walls of the jet channel so that the fluid takes on a rotational motion and is then injected into the space between the two plates. The jet walls have a no-slip boundary condition and we apply the condition of the inlet velocity at the inlet, the information about which is given in table 1.

Table 1: Boundary condition of velocity at the jet entrance (researcher's findings)

speed m/s	16.9
To apply speed	y
temperature (kelvin)	300
Turbulence intensity	% 3.6
Turbulence length scale	0.0033

After passing through its channel, the jet enters the chamber between the two walls, where the fluid velocity is zero and the fluid is at rest.The lower wall of the chamber has a constant flux boundary condition and the numerical value of the flux is equal to 1000 watts per square meter. Table 2 shows the boundary conditions and the initial condition in the jet injection area.

Table 2: Boundary conditions and initial conditions in the jet injection area (researcher's findings).

speed m/s	0
Low level flux (watts per square meter)	1000
Turbulence energy	0
Turbulence energy dissipation rate	0

In order to be able to model the flow and heat transfer in the chamber, it is necessary to solve the equations of continuity, momentum, energy and turbulence and apply the boundary conditions mentioned in the two tables above. The boundary conditions are chosen in such a way that the results can be compared with the previous works for the jet without a groove. The model used here is three-dimensional, which means that the momentum equations must be solved in three dimensions. Also, the equations do not depend on time and the results are reported reliably. Because the jet is moved by grooves, the k-ε RNG model is used, which is used for rotational movements. Also, we model and examine the behavior of the flow near the wall by using the standard wall function.

In the end, it is stated that in obtaining the results, the effect of placing two jets together is also considered and it will be shown how two jets together affect the heat transfer.

First, the independence of the numerical solution from the size of the computing networks has been investigated. Therefore, the effect of network size changes on the Nusselt number has been investigated. The number of computing cells is 256,510, because with this number, there is no change on the average Nusselt number on the hot plate, and the results will be independent of the number of computing cells and the grid size. To check, a jet is located above the center of the hot plate. The jets are positioned so that their center is on the axis. z and the distance of their center from each other is equal to the distance of the center of each of them from the border. In this case, the output flow from the single jet moves directly towards the hot plate and is deflected due to collision with it and flows along the plate towards the outlet of the domain. In this study, it was shown that the diameter of the fluid exiting from the nozzle increases by moving away from the jet opening and approaching the hot plate due to the pressure distribution. If two jets are placed in close proximity to each other, we will have two completely different areas, that is, the area limited between the two jets and the area created between the injection and outlet jets is the computational domain. In the first area, the outflow of each of the two jets is affected by the other and cannot leave the domain without interference. While in the second region, the flow behavior is similar to when a jet is injected into the domain.The outflow from the jet is deflected when it hits the hot plate and then exits the domain exit area. After hitting the hot plate, the output of each jet first changes direction towards each other, then moves upwards and creates two vortices in this place. In four jets, four located on the vertices of the square, the behavior of the flow is the same as before, with the difference that there is a single jet in the area on the axes.

The flow exiting from the jet hits the hot plate without any disturbance and flows along it in areas on the axes that overlap and form vortices. In the area between the four jets, the fluid flow is much more complicated because all the outflows from the jets affect each other and many vortices are created in this area.

Jets are used in this study to cool the hot plate. With one jet, the average Nusselt number is 53.47, with two jets, it is 58.58, and for four jets, this number is 68.12, which increases with the increase in the number of jets.

In order to investigate in more detail, the temperature contour on the hot plate was investigated for different number of jets. When using a jet, the temperature lines are circular concentric lines and the temperature increases away from the center where the jet hits the hot plate. Temperature contours In cases where two and four jets are used for cooling, high temperature regions are formed in the regions where the outflow from the jets affect each other. Therefore, when using two jets for cooling, the maximum temperature on the hot plate is higher than when using one jet.

The distance of the nozzle is closer to the target and therefore more part of the fluid flows on the surface of the target and therefore extracts more heat. Nusselt numbers decrease. At a temperature of 50 and an angle of 90, the Nusselt number is 136.5, and by doubling the distance, this number is 112.3. At a temperature of 50 and an angle of 60, the Nusselt number is 115.7, and by doubling the distance, this number becomes 108.6. At a temperature of 70 and an angle of 90, the Nusselt number is 146.8 and by doubling the distance, this number is 132.4. At a temperature of 70 and an angle of 60, the Nusselt number is 135.1, and by doubling the distance, this number is 121.8.

By using rotating jet, it can be used to change the flow behavior and heat transfer rate. In order to rotate the jet fluid flow, a warped strip is used, which is embedded in the jet pipe with a length of 0.1 m. This strip is twisted at different angles of 180 , 360 and 720. The fluid coming out of the jet nozzle continues to rotate even after leaving the tube where the twisted strip is located, and the rotating flow of the fluid exits after hitting the The hot plate is redirected and moves towards the exit of the computational domain. In this step, the average Nusselt number was calculated again. Using a strip twisted at an angle of 180, the average Nusselt number is 53.84, for a strip twisted at an angle of 360, the average Nusselt number is 52.88, and for a strip twisted at an angle of 720, the average Nusselt number is 51.14, which decreases with the increase of the angle.

The temperature contours on the hot plate have been used to check the maximum and minimum temperature as well as the temperature distribution on the hot plate. The results showed that in all cases, using twisted strips, the maximum temperature as well as the minimum temperature of the hot surface is higher than the case where the non-rotating jet is injected on the hot plate.

In order to investigate the behavior of rotating jets that are placed in the vicinity of two rotating jets in the computational domain. To rotate the outflow of the jet, strips are placed in the tube, which is twisted at angles of 180°, 360° and 720°. It can be seen that by placing two rotating jets next to each other, the Nusselt number where the angle of the twisted strip is equal to 180 is (59.2), increasing and where the angle of the twisted strip is equal to 360 (58.19) and 720 (56.04), the Nusselt number decreases. By using two rotating jets and a twisted strip at different angles, the Nusselt number for the twisted strip increased by 180 and decreased for the other two angles.

Conclusion

In this study, the effects of rotating jets as well as different jet arrangements on flow and heat transfer have been investigated. Unlike most previous researches, instead of creating grooves on the tube body, we used twisted strips to divert the flow. The use of a warped strip has the advantage that a simple jet can be transformed into a rotating jet without major changes in the jet structure. In this research, the numerical analysis of the effects related to the number of jets, the use of rotating jet and the use of twisted tape at different angles in terms of heat transfer rate and average Nusselt number have been discussed and evaluated.

In this study, the Nusselt number is highly dependent on the number of jets and the amount of twisting of the irradiated strip, and the highest Nusselt number is obtained when the strip is twisted at an angle of 180. In this research, the temperature distribution on the hot plate with constant heat flux was investigated.Based on the obtained results, the rotating jet has an adverse effect on the temperature distribution on the hot plate, and the minimum and maximum surface temperatures are higher than non-rotating jets. In this review it was shown:

- The diameter of the fluid outflow from the nozzle increases as it moves away from the jet opening and approaches the hot plate due to the pressure distribution.
- The distance between the nozzle and the target has a significant effect on the heat exchange on the target surface, it corresponds to a lower average wall temperature and higher heat transfer coefficients.
- As the distance of the jet from the plane increases, the Nusselt number decreases. In cases where the jet is very close to the surface, it completely affects the surface, and as the jet moves away from the hot plate, the effects of the outflow from the jet on the plate are reduced and the Nusselt number decreases.
- When using two jets in the vicinity, the output of each jet creates two vortices in this place after hitting the hot plate.
- When four jets are used, the flow from the jet hits the hot plate and creates vortices because all the outflows affect each other.
- As the number of jets increases, the Nusselt number increases.
- As the number of jets increases, the average Nusselt number and the maximum temperature on the hot surface also increase. Therefore, in applications where the temperature level should not be too high at any point, it is better to use one jet instead of multiple jets for cooling.
- Based on the results, the average Nusselt number with the angle of the twisted tape that is placed inside the tube increases at an angle of 180 and decreases at angles of 270 and 360.
- In all cases, using twisted strips, the maximum temperature as well as the minimum temperature of the hot surface is higher than when the non-rotating jet is injected on the hot plate.
- By using two rotating jets and a twisted strip at different angles, the Nusselt number for the twisted strip increased by 180 and decreased for the other two angles.

- With the rotating jet and use of a twisted strip with an angle of 180, the degree of the Nusselt number is increased and therefore the maximum temperature of the hot surface is reduced.

References

[1] J. Barrau, S. Riera, E. Léveillé, L. G. Fréchette and J. I. Rosell, *Nozzle to plate optimization of the jet impingement inlet of a tailored-width microchannel heat exchanger*, Experiment. Thermal Fluid Sci.**67** (2015), 81–87.

[2] R. Fernandez-Feria, E. Sanmiguel-Rojas and E.S. Benilov, *On the origin and structure of a stationary circular hydraulic jump*, Phys. Fluids **31** (2019), no. 7, 072104.

[3] R.J. Goldstein and M.E. Franchett, *Heat transfer from a flat surface to an oblique impinging jet*, J. Heat Transfer **110** (1988), no. 1, 84–90.

[4] A. Ianiro and G. Cardone, *Heat transfer rate and uniformity in multichannel swirling impinging jets*, Appl. Thermal Engin. **49** (2012), 89–98.

[5] K. Jambunathan, E. Lai, M. Moss and B.L. Button, *A review of heat transfer data for single circular jet impingement*, Int. J. Heat Fluid Flow **13** (1999), no. 2, 106–115.

[6] M.F. Koseoglu and S. Baskaya, *The role of jet inlet geometry in impinging jet heat transfer, modeling and experiments*, Int. J. Thermal Sci. **49** (2010), no. 8, 1417-1426.

[7] A.M. Kuraan, S.I. Moldovan and K. Choo, *Heat transfer and hydrodynamics of free water jet impingement at low nozzle-to-plate spacings*, Int. J. Heat Mass Transfer **108** (2017), 2211–2216.

[8] M.D. Le, C.M. Hsu, N. Kholili and S.H. Lu, *Effects of axial jet-to-wall distance on flow behavior and heat transfer of a wall jet at low Reynolds number*, IEEE Int. Conf. Adv. Manufact. (ICAM), 2018, pp. 73–76.

[9] D.H. Lee, J. Song and M. C. Jo, *The effects of nozzle diameter on impinging jet heat transfer and fluid flow*, J. Heat Transfer **126** (2004), no. 4, 554–557.

[10] O. Manca, P. Mesolella, S. Nardini and D. Ricci, *Numerical study of a confined slot impinging jet with nanofluids*, Nanoscale Res. Lett. **6** (2011), no. 1, 1–16.

[11] H. Martin, *Heat and mass transfer between impinging gas jets and solid surfaces*, Adv. Heat Transfer **13** (1977), 1–60.

[12] M. Molana and S. Banooni, *Investigation of heat transfer processes involved liquid impingement jets: a review*, Brazil. J. Chem. Engin. **30** (2013), 413–435.

[13] L. Nakharintr, P. Naphon and S. Wiriyasart, *Effect of jet-plate spacing to jet diameter ratios on nanofluids heat transfer in a mini-channel heat sink*, Int. J. Heat Mass Transfer **116** (2018), 352–361.

[14] K. Nanan, K. Wongcharee, C. Nuntadusit and S. Eiamsa-Ard, *Forced convective heat transfer by swirling impinging jets issuing from nozzles equipped with twisted tapes*, Int. Commun. Heat Mass Transfer **39** (2012), no. 6, 844–852.

[15] P. Naphon, L. Nakharintr and S. Wiriyasart, *Continuous nanofluids jet impingement heat transfer and flow in a micro-channel heat sink*, Int. J. Heat Mass Transfer **126** (2018), 924–932.

[16] H. Samma, A. Khosrojerdi, M. Rostam-Abadi, M. Mehraein, and Y. Cataño-Lopera, *Numerical simulation of scour and flow field over movable bed induced by a submerged wall jet*, J. Hydroinform. **22** (2020), no. 2, 385–401.

[17] B. Weigand and S. Spring, *Multiple jet impingement*, TURBINE-09. Proc. Int. Symp. Heat Transfer in Gas Turbine Syst., Begel House Inc., 2009.

[18] P. Xu, B. Yu, S. Qiu, H. J. Poh and A.S. Mujumdar, *Turbulent impinging jet heat transfer enhancement due to intermittent pulsation*, Int J Thermal Sci **49** (2010), no. 7, 1247–1252.

[19] Y.T. Yang and F.H. Lai, *Numerical study of heat transfer enhancement with the use of nanofluids in radial flow cooling system*, Int Jo Heat Mass Transfer **53** (2010), no. 25-26, 5895–5904.