International Journal of Innovative Research and Reviews, 2(2):1-9

International Journal of Innovative Research and Reviews

ISSN: 2636-8919

Website: www.injirr.com

doi:

Research paper, Short communication, Review, Technical paper

RESEARCH ARTICLE

An Investigation of the Effect of Vortex Generators on Heat Transfer in Channel Flow

Sinan ÇALIŞKAN[1], Muammer ZIRZAKIRAN[2,*]

[1] *Hitit University, Faculty of Engineering, Department of Mechanical Engineering, Çorum, TURKEY*
[2] *Atatürk University, Erzurum Vocational School, Erzurum, TURKEY*

**Corresponding author E-mail: mamozir@atauni.edu.tr*

HIGHLIGHTS

> The influence of the winglet, heat transfer, and pressure drop on the vortex flow to create a vortex in a stream of air that sent to a fixed channel through a fan.

> The increases in heat transfer of the winglet types have been examined and discussed.

ARTICLE INFO

Received : 10.08.2018
Accepted : 10.24.2018
Published : 12.15.2018

Keywords:
Heat transfer,
Vortex generator,
Winglet

ABSTRACT

In this experimental study, we investigated the influence of the winglet, heat transfer, and pressure drop on the vortex flow to create a vortex in a stream of air that sent to a fixed channel through a fan. Three parameters were determined as the winglet parameters: angle, height of the winglet, and arrangement in the channel of the winglet. The three winglet placed in the channel have the same angle and the same base height. Its experimental set-up was applied for four different Reynolds numbers. The Nusselt number obtained from the smooth channel was compared with the correlations in the literature. Heat transfer results were obtained using thermal imaging technique. The heat transfer data obtained for all the specified test cases were compared with the empty channel data. While comparing the Nusselt numbers and thermal camera images obtained from the experiments, the increases in heat transfer of the winglet types have been examined and discussed.

Contents

1. Introduction

Nowadays, the increasing heat transfer enhancement and therefore consequently the saving material and energy saving have gained importance. The heat transfer enhancement in industrial flow systems is a phenomenon that needs to be continuously developed. In many engineering applications, heat is generated by the operation of the system. If the heat is not removed, the system may cause overheating problems and damage to the system. In recent years, the studies in the field of cooling of electronic devices, automotive and space vehicles, nuclear reactors and heat exchangers have gained speed. Moreover, the gas turbines also have the same situation. In order to increase the thermal efficiency of the gas turbines, it is desirable that the gas temperature at the inlet is to be high. However, in order to overcome with the super-

Cite this article Çalıskan S, Zırzakıran M. An Investigation of the Effect of Vortex Generators on Heat Transfer in Channel Flow. *International Journal of Innovative Research and Reviews (INJIRR)* (2018) 2(2):1-9

Link to this article: http://www.injirr.com/articleview/19

heat fluxes and thermal stresses caused by hot gas, the heat must be effectively removed from the winglet, so that the superheating does not shorten its service life. The cooling process is primarily carried out in two ways, namely liquid and gas cooling. In liquid cooling, high thermal conductivity is preferred in industry due to high specific heat capacities.

Hemmat Esfe et al. (2015), in their numerical study, quantitatively examined the heat transfer and friction factor changes due to different parameters such as obstacle sequence and obstacle geometries by passing the fluid through the channel. Depending on flow and geometry, heat transfer and friction characteristics were developed using finite volumes and SIMPLE algorithm to develop temperature dependent and temperature independent correlations. The data show that the obstacles used increase the Nusselt number by 10% less than the average Nusselt number [1].

Tanda (2011) examined the effect of obstacle intervals on heat transfer and thermal efficiency at the 9000-35500 Reynolds values by placing obstacles at an angle of 45° into the rectangular channel. At the end of the study, the best performance was obtained for p / e = 13.3 [2].

Alamgholilou and Esmaeilzadeh (2012) investigated the effect of heat transfer under laminar flow (500-2000) and turbulent flow (2000-4500) by placing obstacles sequentially into the rectangular section channel. In this study, hydrodynamic and thermal behaviors of the flow were investigated by passive, active and compound methods. They made measurements according to the ratio of the gap between obstacles (S) to the height of the obstacle. The best thermal performance was measured as 6% - 13% in the study using the passive method under high Reynolds numbers [3].

Chatterjee (2012) examined the effect of the vortex factor behind the obstacles on the thermal surfaces by using square and cylinder obstacles in the channel at very low Reynolds numbers (10-40). The different characteristics of the flow area have been measured according to the Reynolds number, the shape of the barrier and the blockage parameter. In the numerical study performed, the aim is to estimate the critical value of the thermal flow parameter (Richardson number) for the beginning of vortex formation [4].

Agrawal et al. (2015) examined the transfer of heat transfer by placing obstacles on the channel wall. They have made measurements of the unhindered and disability of the wall surface in different regions of the channel with the help of laser. According to the results, the efficiency of the heat transfer on the wall surface which is obstructed was found to be higher than the measured temperature values on the flat wall surface by 4% and 7%. As a result of the experimental study, they determined that there was a decrease in heat transfer in areas where obstacles were close to the base [5].

Dogan et al. (2006) investigated the effect of heat transfer on the upper and lower surfaces of the channel. In the experimental study, various Reynolds and Grashof numbers and AR = 2,4,10 were used parametrically. Measurements mean surface temperature and distribution of Nusselt numbers were obtained and the effects of Reynolds and Grashof numbers were evaluated. As a result of the experimental study, it was determined that the increase in the number of Grashof increased thermal efficiency. When the results were synthesized according to the AR ratios, the upper heater determined that the average surface temperatures were higher than the average surface temperature of the lower heaters in the case of AR = 2. It was observed that Nusselt number had a positive effect on heat transfer development for Reynolds numbers [6].

Gül et al. (2006) examined the effect of a square cross-section obstacle placed on the channel base perpendicular to the flow in a rectangular cross-section and placed on the heat transfer. Reynolds number between 3000 and 15000, horizontal and vertical direction of the position of the change of the obstacle and heat transfer on the size of the effective and maximum heat transfer is provided to obtain the optimum parameters. As a result of the study, they achieved a 142% improvement in heat transfer, but they determined that there was a 200% increase in friction losses [7].

Meinders and Hanjalic (2002) examined the effect of the heat transfer of the position of the obstacles in the channel with turbulent flow conditions. In this study, local heat transfer has been found to be significantly changed according to the arrangement and location of the placed obstacles. According to the obtained data, the heat transfer coefficient obtained for the obstacles placed in the channel was determined to be independent from the location where the obstacle was placed [8].

Kıvılcım (2007) experimentally examined the effect of non-circular rotational barriers which are positioned perpendicular to the flow in a horizontal rectangular channel. Reynolds number is examined in the 3000-15000 range, square and circular cross-section in two different geometries and each of three different sizes of the selected obstacle in the horizontal and vertical direction in nine different locations by changing each obstacle in both fixed and rotating experiments have done. As a result of the measurements, it was found that the square section barrier provided a better heat transfer than the circular cross-section barrier. In addition, the increase in the size of the obstacle increased in direct proportion to heat transfer [9].

Xie et al. (2014) examined the effects of rib cross-sections on heat transfer and their cooling performances using the Fluent program. In their study, they used obstacles with six different internal cross-sections and positioned these obstacles at the center of the channel at 45, 90 and 135^0. In their investigation, they took the Reynolds number between 10000 and 50000. The highest heat performance is provided in the case where the mid-section is positioned at 135^0. In addition, it was found that better heat transfer was achieved in cases where Reynolds number is high [10].

Anghel and Anglart (2012) examined the effect of heat transfer on experimentally by passing high pressure water through the system by using various obstacles in vertical pipes. In their work, they used five different experimental systems. In the first case, 2 mm circular cross-section obstacle, in the second case circular cross-sectional obstacle and rectangular cross-section obstacle in the other case with caged structure barriers were placed experimental measurements were made. Measurements were made with 88 thermocouples connected to the system. As a result of the investigations, it was observed that the obstacles used in the pipes greatly affected the flow. Improvement of critical heat flux is achieved between 6% and 17% in cases where the cylindrical and truss-section barrier is used [11].

Frohlich et al. (1988) in their numerical study, the effect of heat transfer on a circular cross-section with a rectangular section perpendicular to the flow was investigated using the LES method. The Reynolds number was 3900 and 140000. They compared their measurements with the values found in different studies in previous studies [12].

Bhadouriya et al. (2015) examined the effects of air flow on the heat transfer and friction factor in the bent square section duct in their numerical study. The experiments were carried out with air between 11 and 16.5 values and the Reynolds number between 600 and 70000. As a result of the studies, better heat transfer and pressure drop were obtained in the channel having a value of 11.5 bends compared to the flat square channel. The results obtained were stated to help compact heat exchanger designs.

Ahmed et al. (2015), in their numerical and experimental study, using heat exchanger and fluid properties in equilateral triangular channels using combined vortex producers and nano fluids. They suspended two different nano fluids in water and passed through the channel. As a result of the experimental and numerical studies performed, it was observed that the use of vortex producers in the channel with the base fluid had a positive effect on heat transfer and thermal efficiency. As a result of the measurements, it has been observed that the reduction in pressure can be achieved by using compound vortex manufacturers [14].

Gutierrez et al. (2015) examined the effect of different geometries in parallel with rectangular cross-section on obstructions, block heaters and curvilinear current limiters and examined the effect of heat transfer on experimentally and numerically. In the study, the purpose of the deflators is to increase the heat movement within the channel and to change the flow between the blocks. The general aim of the study is to determine the effect of parameters such as obstacle geometry, deflector position, channel height and deflector radius for minimum pressure drop and maximum heat transfer. As a result of the studies, it was found that different geometric factors were a strong factor on heat transfer. The best thermal efficiency was measured in the case of a single deflector [15].

Gül and Evin (2006), in their experimental study, examined the effect of section change on flow characteristics. In their study, they observed the distribution of horizontal and vertical velocities by performing a section change along the 300 mm channel. They determined the cross section ratio of the channel by the channel axes of the upper and lower edges of the channel and the side edges of the channel as 4° to 5°, respectively. Reynolds number in the range of 12000-50000. As a result of the study, due to the increase in the area along the axis in the channel, the decreases in speed as the fluid progresses in the channel are clearly seen. It was determined that the cross-sectional area was effective on the velocity distribution of the flow [16].

Chang et al. (2010) investigated the effect of the walls on the heat transfer by using rough and dimpled hexagonal channels. They used 4 different pin-fin geometries and the Reynolds number in the range of 900-30000. The best re sult was the convex-concave wall channel [17].

Chaitanya and Dhiman (2012) examined the effect of heat transfer and friction factor by solving two momentum and energy equations by placing two obstacles in circular cross-section. They then analyzed the same situations in the ANSYS Fluent program. The Reynolds number was determined as 1-40, Prandtl number was 50 and T / D ratios were determined as 1.5-4.0. In the light of the data obtained, it was determined that the average Nusselt number increased by increasing the T / D ratio. The best thermal efficiency and low coefficient of friction were 42% for the parameters Re = 40 and T / D = 1.5 [18].

Chen et al. (2015) examined a change in the heat flow and vortex formation on the cylinder by passive flow control method by placing a pipe with a circular cross section through which holes were drilled into a wind tunnel. In their study, Reynolds number was 41600 and circular obstacle size was 70 mm, outer diameter was 78 mm and thickness was 30 mm. On the circular obstacle, they opened channels on 24 pieces of 15x4 mm. They made their measurements according to different X / D ratios. As a result of the investigations, using passive jet control method, they achieved improvement of 33.7% and 90.6% in the flow [19].

Evin and Tanyıldızı (2006), in their experimental study, studied the effects of circular barriers of different diameters placed in a direction perpendicular to the flow to the upper flow region in a partially heated rectangular channel. Depending on the measurements made, it was observed that the average Nusselt numbers reached the maximum values at certain distances from the obstacle, horizontal and vertical. It is determined that the data obtained in the channels that are used in obstacles are higher than the values obtained in the empty channel. In comparison to the unobstructed situation, 1.35 times the barrier with 0.03 m diameter and 1.2 times higher heat transfer with 0.02 m diameter obstacle has been obtained [20].

Selimefendigil and Öztop (2014) examined the effect of a rotating obstacle placed on the heat transfer in the channel with the numerical study. They placed a magnetic system on the bottom surface of the channel and created a magnetic effect on the barrier. In their study, they took Reynolds number between 10 and 200 and obstacle rotation angle between -75 and 75 degrees. In the light of the data obtained from the study, it was determined that the number of local Nusselt numbers increased by decreasing the magnetic effect and increasing the Reynolds number [21].

Barik et al. (2015) examined the hydrodynamic changes and thermal changes in the rectangular duct with small cross-sections with different protruding surfaces. They used three different protruding surfaces, rectangular, triangular and trapezoidal. According to the data obtained, more heat transfer and development rate is obtained compared to the data obtained with triangular protruding channel compared to the other projection geometries [22].

Hussam et al. (2015) investigated the flow of heat by placing circular obstacles in a rectangular channel. They considered different Reynolds numbers and the height of the barrier from the base of the channel. They determined G / D ratios as 4.5, 2.25, 1.13, 0.75, 0.38 and 0.19. As a result of the investigations, it has been observed that the distance of the obstacle from the base surface changes greatly the thermal efficiency and flow. A 48% improvement in heat transfer was detected when the circular obstacle was placed in the center of the channel [23].

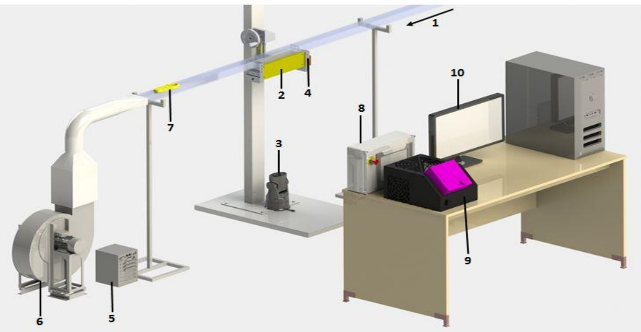
Baytaş et al. (2011), in their experimental study, they put a square, square and 45° rotated square barriers into a channel with rectangular cross-section and examined the effects on the heat transfer by creating porous ambient conditions. For these three cases, velocity distributions, interface and Darcy velocities in the channel section were determined experimentally. According to the data obtained from the study, it was found that the maximum speed was the square obstacle. The lowest maximum velocity was found to occur in the case of using square obstacles rotated 45^0 [24].

Ling et al. (1994) investigated the heat transfer and pressure drop experimentally in a square sectioned channel with fixed surface temperatures, obstacles on the lower and upper surfaces with cross-section triangular triangles. found that the heat transfer in the channel using triangular cross-sections with obstacle heights and steps at different Re numbers increased by 1/2.3 according to the heat transfer in the smooth surface channel, whereas in the pressure drop there was an increase by 1/10 [25].

Huq and Aziz-ul Huq (1998), uniform turbulent flow in a diffuser pipe on the inner surface of the peripheral axial heat transfer characteristics experimentally examined eight as in the studies that have heat transfer coefficient based on wingless pipe flow committed to increasing at a rate of 97-112%.

2. Material and Method

The experimental apparatus used in this study is shown in detail in Figure 1. In this system, fan, air inlet section, flow regulator, thermal camera and flow rate, pressure difference and temperature measuring devices are available. The air flows through the test section of the duct by a fan of variable speed. The total length of the channel is 2500 mm. All channels used in the experimental study are made of 5 mm thick Plexiglas material.



1. Flow direction	6. Fan
2. Test section	7. Anemometer
3. Thermal camera	8. DC power supply
4. Pressure probes	9. Micro-manometer
5. Fan speed adjuster	10. Computer

Figure 1 Overview of experimental layout

The heater plate is 277 mm length and 100 mm width. The heater surface used in the experiments is made of stainless steel foil and is properly stretched between the two copper plates. To create a continuously heated flow surface, the foil is heated by using a DC power supply. Thermal images are obtained with a thermal camera positioned vertically under the heater. The mean heat transfer coefficient in the stainless steel foil surface was measured for variable air flow rates inside the duct.

2.1. Test section

Stainless steel foil has been used as internal heating surface to provide constant heat flux on the surface, as seen in Figure 2. The lower surface of the 0.02 mm thick stainless steel foil is coated with 0.01 mm thick black paint layer to calculate the losses that will occur with radiation on the heating surface (stainless steel foil). In this experimental study, the emissivity values for painted and unpainted surfaces were 0.82 and 0.13, respectively, because the same properties were used as foil. As a result of the research, 0.02 mm thick stainless steel foil is very thin, the temperature of the top surface of the stainless steel foil is equal to the temperature of the bottom surface was determined. Therefore, in our experimental study, the heat transfer calculations were performed by equalizing the upper surface and lower surface temperatures of stainless steel foil.

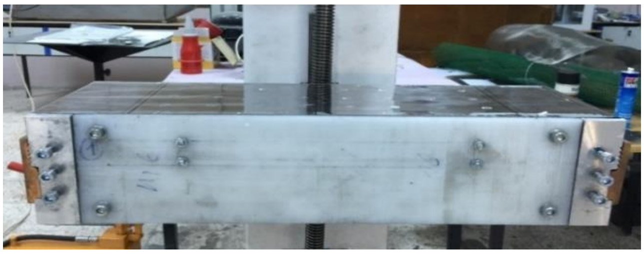


Figure 2 General appearance of test section

2.2. Winglet

The winglets are made of 1 mm aluminum. The winglets used in the study are given in Figure 3. As can be seen from Figure 3, the winglets were used as a vortex generator by opening from the base distances at certain heights.

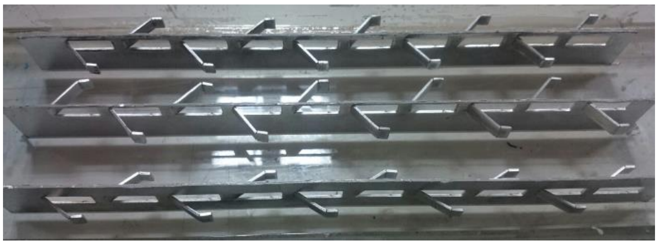


Figure 3 The winglet with 3 mm in height from the base and 30^0 angle.

The proper stretching of the heating surface is very important for us to achieve more realistic and clearer results than our experimental work. The stainless steel foil is heated with the help of DC power supply which can be changed in 0-6 V and 0-110 A to provide a constant heat flux on the heating surface. The DC power supply is connected to the heating surface from six different points with copper bur in the test section. In this way, it is possible to give power to the heating surface at different current and voltage values.

In our experimental system, FLIR A640 thermal camera was used to determine the temperature distributions on the heating surface. Thermal images on stainless steel foil are obtained by placing them perpendicular to Z in the lower side of the test section of the thermal camera. The camera can measure temperatures from -20 to 1200°C with approximately ±2% accuracy. Thermal camera system we use, 7.5 to 13 micrometers between 320 to 240 pixels are uncooled focal surface determination. The field of view is 25x18,8°/0.4 and the instant field of view is 1.3 m-rad and thermal sensitivity is 0.07^0C – 30^0C. The images obtained with the thermal camera were then viewed and recorded for analysis using the FLIR-Quick-Report program with the help of the computer.

The air is supplied to the test system through a fan with a capacity of 2950m^3/h, which can be set to cycle. In order to achieve different speeds, the fan flow must be adjusted at different values. The most efficient and effective method for adjusting fan flow is the use of frequency control devices. The frequency control device operates in the range of 0-50 Hz and has a sensitivity of 0.01 Hz. In the experiments performed, LV-107 model anemometer was used for speed measurements. The temperature at the inlet and outlet of the test section was measured using a K-type digital thermometer.

2.3. Data Analysis

The amount of heat transfer by convection from the heating surface is expressed by the following equation:

$$Q_{conv} = Q_{el} - Q_{loss} \tag{1}$$

Heat transfer is defined as Q_{conv} by convection occurring at the lower side of stainless steel foil and conduction occurring on side surfaces. The electrical power supplied to the system is described in Q_{el} as follows.

$$Q_{el} = VI \tag{2}$$

In this equation, V refers to the voltage given to the system and I refer to the current given to the system.

Radiation heat flux q_r occurring on both surfaces of the heating surface:

$$q_r^{\text{front}} = \varepsilon_t \sigma (T^4 - T_b^4) \tag{3}$$

$$q_r^{back} = \varepsilon_b \sigma (T^4 - T_\infty^4) \tag{4}$$

ε_b and ε_t are the emissivity rate of painted and unpainted surfaces, respectively.

Stefan-Boltzmann constant, T_b and T_∞ are surface and air temperatures, respectively.

Heat transfer from the natural convection under the plate is calculated using the following formulation:

$$q_f = h_f (T - T_\infty) \tag{5}$$

Where h_f is the free convection coefficient defined as 1.1 W/m^2K for 0.1 m/s air velocity.

The heat conduction is calculated by the equation given below:

$$q_c = k \frac{\Delta T}{t} \tag{6}$$

Here, k is the thermal conductivity of stainless steel foil. ΔT is the temperature difference across the stainless steel foil, and T is the thickness of the stainless steel foil.

T and $T_{b,x}$ represent the film temperature of the heater surface and the fluid, respectively.

The Nusselt number was calculated based on the hydraulic diameter (D_h) of the rectangular channel. In this equation, h_x is the coefficient of convection and k is the coefficient of thermal conductivity of air.

The average Nusselt number (Nu_m) is calculated as follows:

$$Nu_m = \frac{1}{L} \int Nu_x \partial x \tag{7}$$

The heat flux obtained by convection is evaluated by the equation given below:

$$q_{conv} = \frac{Q_{el} - Q_{loss}}{A_P} \tag{8}$$

In the calculation of the heat transfer surface area equations, W and L terms represent the width of the heating surface and the length of the heating surface, respectively.

The Reynolds number calculated based on the channel hydraulic diameter is given below:

$$Re = \frac{\rho U Dh}{\mu} \tag{9}$$

where D_h is the hydraulic diameter, U is the speed of the fluid, ρ is the density of the fluid and μ is the viscosity of the fluid.

3. Results and Discussion

Experimental data for heat transfer with forced convection were analyzed under different Reynolds number values. The Nusselt numbers are especially calculated for the empty channel by using the experimental results. The Nusselt number obtained for empty channels was compared with the correlation found by Dittus-Boelter under turbulent flow in the literature.

Dittus-Boelter correlation is expressed as follows.

$$Nu = 0{,}023 Re^{0{,}8} Pr^{0{,}4} \tag{10}$$

In Figure 4, the Nusselt number is compared with the correlation given in equation 10. The Nusselt number was found to be agree with the correlation between ±1.5% deviation range.

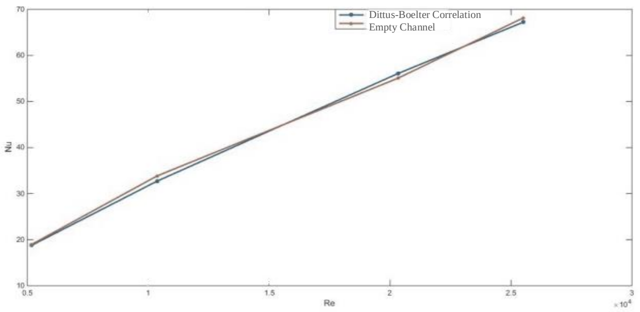


Figure 4 The validation of Nusselt number for empty channel

For the winglets with different geometries, the images obtained with the help of thermal camera to create the maximum field of view, using the FLIR-Quick Report program, the temperature distributions on the surface were obtained and the following images were given. The names and abbreviations used in the study are given in Table 1.

Table 1 Test names used in the study

Experiment Names	Abbreviation	Reynolds number	Abbreviation
Empty channel	X	5181	X1
		10363	X2
		20328	X3
		25510	X4
Height of the winlet separation from the ground =1 mm, winglet angle= 15^0	A	5181	A1
		10363	A2
		20328	A3
		25510	A4
Height of the winglet separation from the ground = 3 mm, winglet angle = 30^0	B	5181	B1
		10363	B2
		20328	B3
		25510	B4
Height of the winglet separation from the ground = 5 mm, winglet angle = 75^0	C	5181	C1
		10363	C2
		20328	C3
		25510	C4

The contours of the temperature distributions of Re = 5181, Re=10363, Re=20328 and Re=25510 for the arrangements of the winglet with different geometry type in the channel along the test region from the test zone entrance and exit points are shown in Figure 5-Figure 8. In our experimental studies, as seen in all the temperature contours, the temperature values in the flow direction starting from the entrance zone are increasing gradually. As can be seen from the temperature contours, the thermal boundary layer begins to develop with the transfer of heat from the hot fluid resulting from the heating of the cold fluid at the inlet, and the thermal boundary layer begins to develop and reaches thermally developed conditions as it travels along the channel. Thermal boundary layer is defined as the regions with temperature gradients and heat transfer. The gradients and their associated heat transfer are neglected on the outside of these areas. Also, this can be clearly shown in the color changes in the temperature contours.

When the data obtained from the temperature contours were examined, it was found that the temperatures at the channel entrance were low. However, after contact of the fluid with the winglet, it has been determined that the temperature increases along the dimensionless x distance and the non-dimensional y distances due to the vortex formation and the increase of the fluid mixture. A better heat transfer on the surface as well as a better heat transfer at the blade angle of 45° was obtained as the distance of the winglet separation from the base distance decreased.

As seen from the Figure 5-Figure 8., when the temperature contours are observed, there are less temperature differences in high Reynolds numbers. This is due to the increase in the Reynolds number, the thermal boundary layer thickness is decreased, the fluctuations caused by the vanes, the heat transfer coefficient increased and the heat transfer increased.

In the experimental study, the primary aim of using different base height and different angle is to increase the heat transfer and heat transfer surface by providing turbulence formation. As can be seen from the temperature contours, the heat transfer in all experimental cases has increased significantly compared to the empty channel.

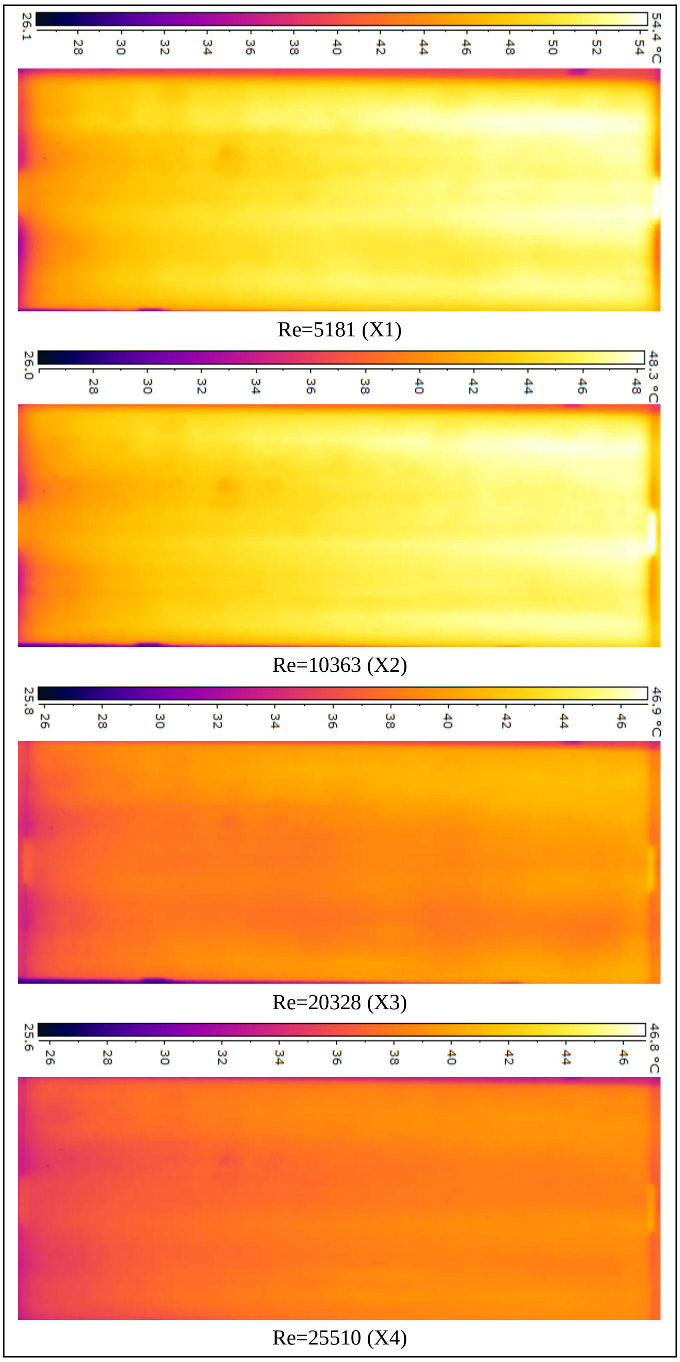
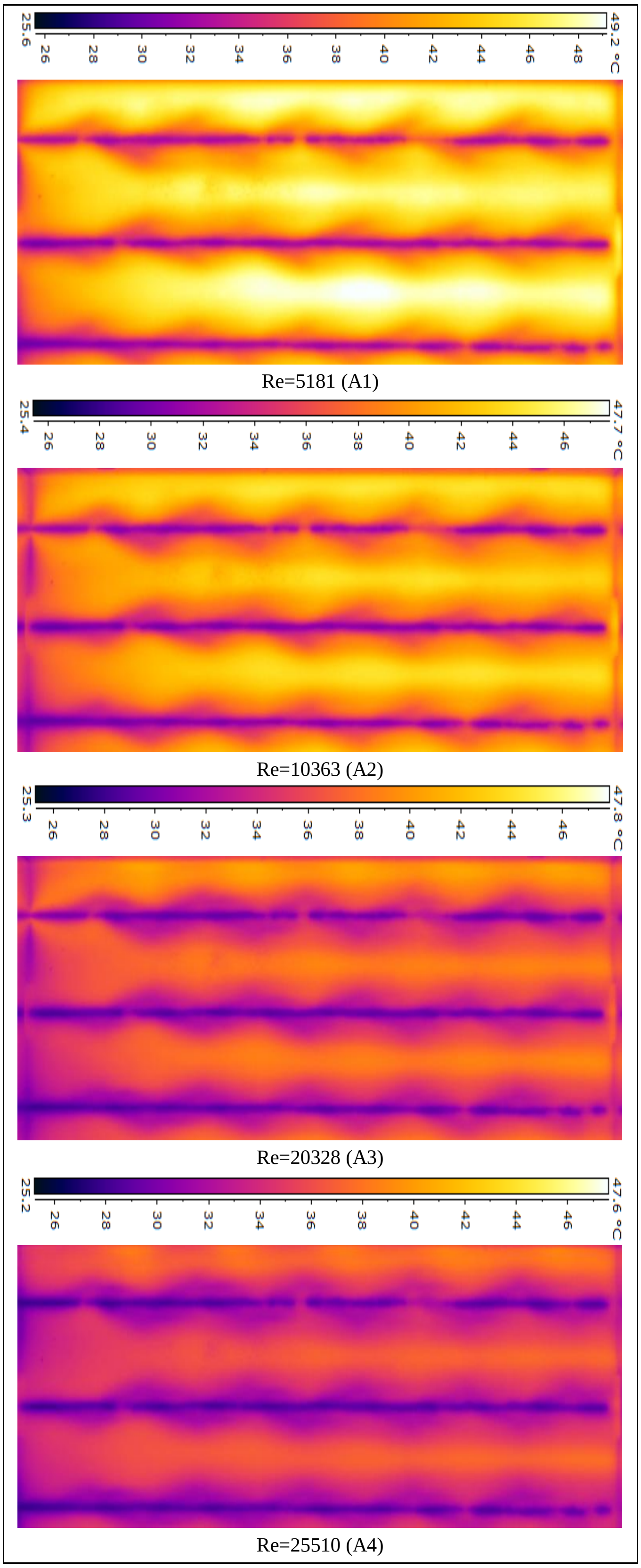


Figure 5 Temperature contours in the empty channel obtained in different Reynolds numbers



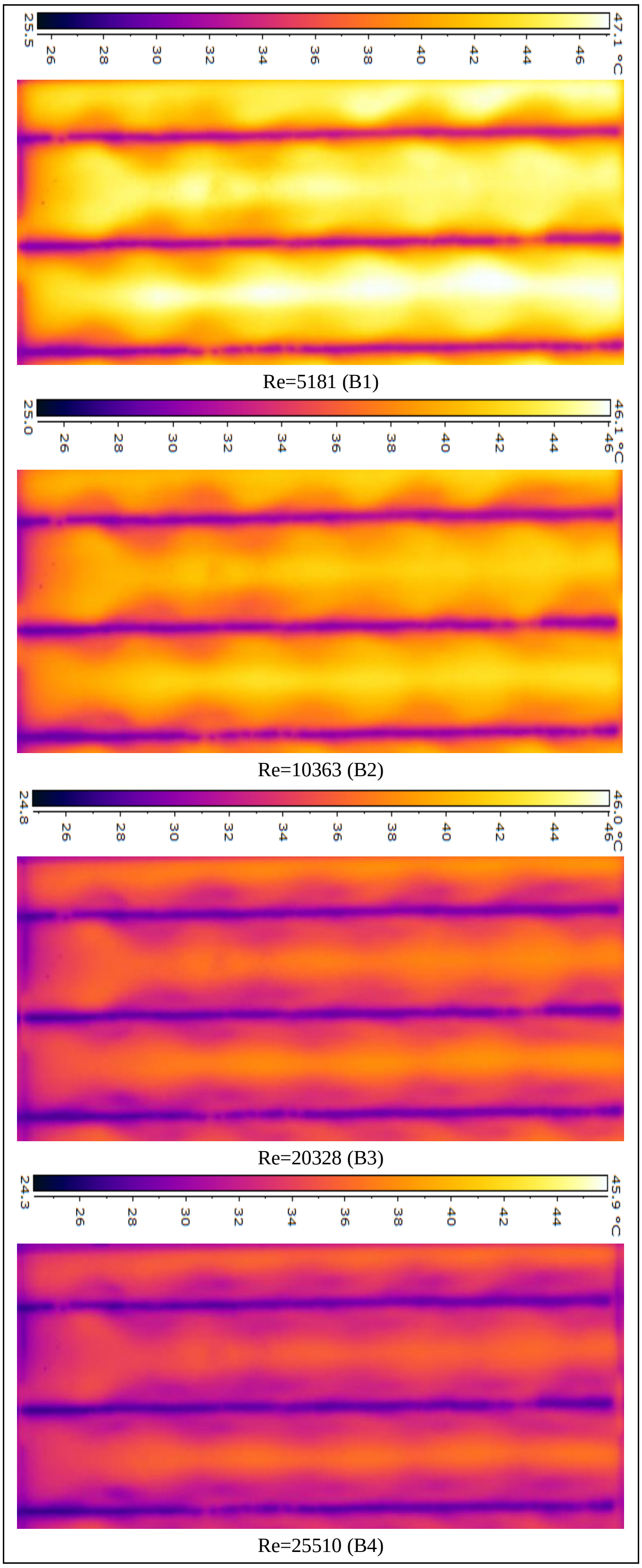


Figure 6 Temperature contours in the winglet channel obtained in different Reynolds numbers (A)

Figure 7 Temperature contours in the winglet channel obtained in different Reynolds numbers (B)

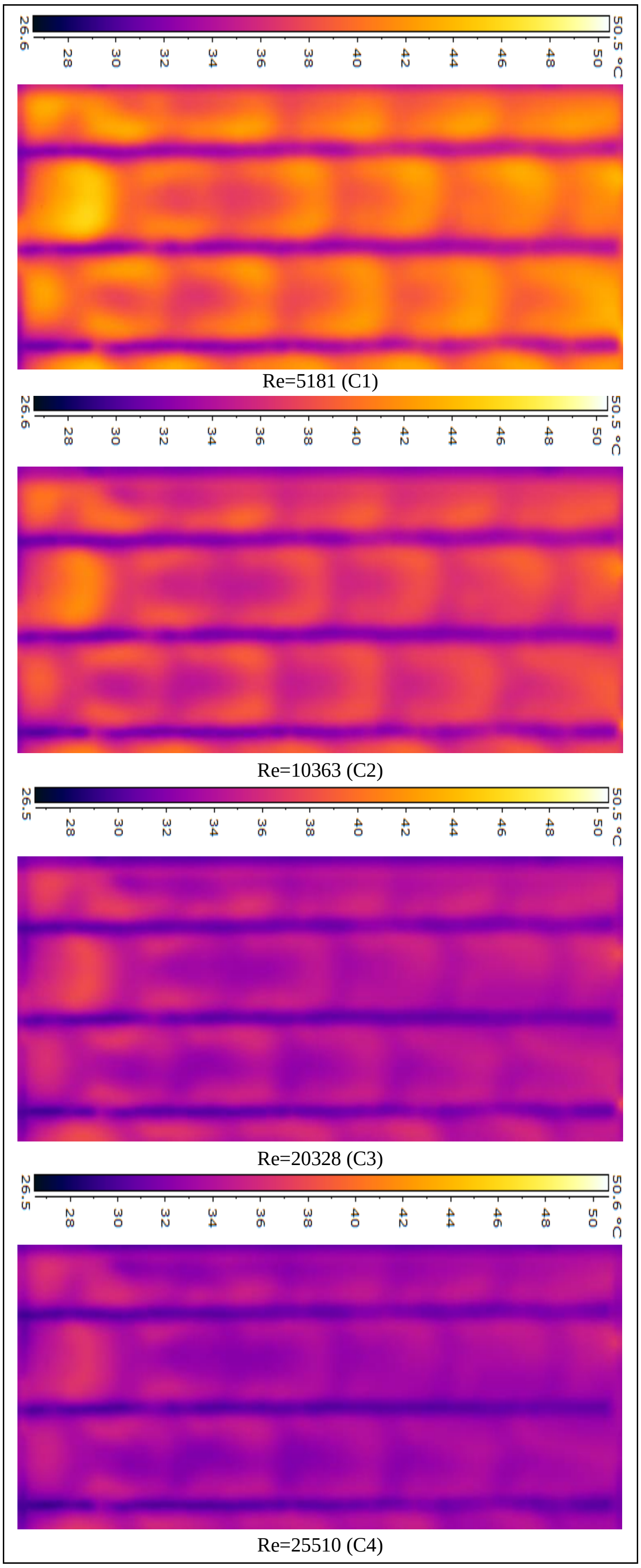


Figure 8 Temperature contours in the winglet channel obtained in different Reynolds numbers (C)

4. Conclusions

1- In the flow and heat transfer analysis, the experiments were performed in the empty channel for the calibration of the experimental set-up and the results were compared with the correlations (Dittus-Boelter) in the literature. Then the heat transfer results for different winglet types were investigated. In heat transfer analysis, the relationship between Reynolds number and average Nusselt numbers for all winglet geometry types were investigated. These investigations examined the angle of the winglet and the effects of the distances of the winglet from the bottom.

2- The Reynolds number of increased for all experimental cases and Nusselt number increased. The highest heat transfer increase was obtained when the winglet was lower than the base distance and the attack angle was 45^0.

3-Maximum increase in Nusselt number for all experiments using a 1 mm base height; the highest Reynolds number (Re = 25510), the attack angle was 45^0, it increased to 1.92 percent compared to the empty channel.

Nomenclature

A	Total area
D_h	Hydraulic diameter
g	Height of the winglet
h	Heat transfer coefficient
H	Channel height
I	Current
k	Thermal conductivity
L	Test region length
Nu	Nusselt number
Nu_{ort}	Average Nusselt number
Q	Heat transfer
Re	Reynolds number
T	Temperature
U	Velocity
V	Voltage
W	Channel width
ΔP	Pressure change
v	Kinematic viscosity
ρ	Fluid density
P	Pressure

References

[1] Hemmat Esfe M, Abbasian Arani AA, Niroumand AH, Yan W-M, Karimipour A. Mixed convection heat transfer from surface-mounted block heat sources in a horizontal channel with nanofluids. *International Journal of Heat and Mass Transfer* (2015) **89**:783–791. doi:10.1016/j.ijheatmasstransfer.2015.05.100.

[2] Tanda G. Effect of rib spacing on heat transfer and friction in a rectangular channel with 45° angled rib turbulators on one/two walls. *International Journal of Heat and Mass Transfer* (2011) **54**(5-6):1081–1090. doi:10.1016/j.ijheatmasstransfer.2010.11.015.

[3] Alamgholilou A, Esmaeilzadeh E. Experimental investigation on hydrodynamics and heat transfer of fluid flow into channel for cooling of rectangular ribs by passive and EHD active enhancement methods. *Experimental Thermal and Fluid Science* (2012) **38**:61–73. doi:10.1016/j.expthermflusci.2011.11.008.

[4] Chatterjee D. Triggering vortex shedding by superimposed thermal Buoyancy around bluff obstacles in cross-flow at low Reynolds numbers. *Numerical Heat Transfer Applied* (2012) **61**:800–806.

[5] Agrawal S, Simon TW, North M, Bissell D, Cui T. Heat transfer augmentation of a channel flow by active agitation and surface mounted cylindrical pin fins. *International Journal of Heat and Mass Transfer* (2015) **87**:557–567. doi:10.1016/j.ijheatmasstransfer.2015.04.036.

[6] Dogan A, Sivrioglu M, Baskaya S. Investigation of mixed convection heat transfer in a horizontal channel with discrete heat sources at the top and at the bottom. *International Journal of Heat and Mass Transfer* (2006) **49**(15-16):2652–2662. doi:10.1016/j.ijheatmasstransfer.2006.01.005.

[7] Gül H, Evin D, Tanyıldızı V. Experimantally investigation of the effect of a square cross-sectional obstacle located in a channel on heat transfer. *Journal of Engineering and Natural Sciences, Sigma* (2006) **3**.

[8] Meinders ER, Hanjalic K. Experimental heat transfer from in-line and staggered configurations of two wall-mounted cubes. *International Journal of Heat and Mass Transfer* (2002) **45**:465–482.

[9] Kıvılcım G. *Yatay ve Dikdörtgen Kanallarda Akış ve Dik Dairesel Olmayan Dönel Engellerin Isı Transferine Etkisi [The Effect of Flow and Non-Circular Rotational Obstacles on Heat Transfer in Horizontal and Rectangular Channels]*. Master Thesis. Sakarya University. Sakarya (2007).

[10] Xie G, Liu J, M. Ligrani P, Sunden B. Flow structure and heat transfer in a square passage with offset mid-truncated ribs. *International Journal of Heat and Mass Transfer* (2014) **71**:44–56. doi:10.1016/j.ijheatmasstransfer.2013.12.005.

[11] Anghel IG, Anglart H. Post-dryout heat transfer to high-pressure water flowing upward in vertical channels with various flow obstacles. *International Journal of Heat and Mass Transfer* (2012) **55**(25-26):8020–8031. doi:10.1016/j.ijheatmasstransfer.2012.08.036.

[12] Frohlich J, Rodi W, Kessler P, Parpais S, Bertoglio JP, Lau-rence D. Large eddy simulation of flowaround circu-lar cylinders on structured and unstructured grids. *Notes on Numerical Fluid Mechanics* (1988):319–338.

[13] Bhadouriya R, Agrawal A, Prabhu SV. Experimental and numerical study of fluid flow and heat transfer in a twisted square duct. *International Journal of Heat and Mass Transfer* (2015) **82**:143–158. doi:10.1016/j.ijheatmasstransfer.2014.11.054.

[14] Ahmed HE, Ahmed MI, Yusoff MZ. Heat transfer enhancement in a triangular duct using compound nanofluids and turbulators. *Applied Thermal Engineering* (2015) **91**:191–201. doi:10.1016/j.applthermaleng.2015.07.061.

[15] Gutierrez LD, Guerrero HA, Ortiz LL, Leon CJ. Numerical and experimental analysis of heat transfer enhancement in grooved channel with curved flow deflectors. *Applied Thermal Engineering* (2015) **75**:800–808.

[16] Gül H, Evin D. Experimental investigation of cross-sectional area effect on characterictics (2006).

[17] Chang SW, Chiang KF, Chou TC. Heat transfer and pressure drop in hexagonal ducts with surface dimples. *Experimental Thermal and Fluid Science* (2010) **34**(8):1172–1181. doi:10.1016/j.expthermflusci.2010.04.006.

[18] Chaitanya NSK, Dhiman AK. Non-Newtonian power-law flow and heat transfer across a pair of side-by-side circular cylinder. *Internationnal Journal of Heat and Mass Transfer* (2012) **55**:5941–5958.

[19] Chen W-L, Gao D-L, Yuan W-Y, Li H, Hu H. Passive jet control of flow around a circular cylinder. *Experiments in Fluids* (2015) **56**(11):848. doi:10.1007/s00348-015-2077-5.

[20] Evin D, Tanyıldızı V. The effect of obstacles normal to the flow in a partially bottom heated horizantal channel on a heat transfer. *Science and Engineering Journal of Fırat University* (2006) **18**:249–255.

[21] Selimefendigil F, Öztop HF. Effect of a rotating cylinder in forced convection of ferrofluid over a backward facing step. *International Journal of Heat and Mass Transfer* (2014) **71**:142–148. doi:10.1016/j.ijheatmasstransfer.2013.12.042.

[22] Barik AK, Mukherjee A, Patro P. Heat transfer enhancement from a small rectangular channel with different surface protrusions by a turbulent cross flow jet. *International Journal of Thermal Sciences* (2015) **98**:32–41. doi:10.1016/j.ijthermalsci.2015.07.003.

[23] Hussam WK, Sheard J, G. Heat transfer in a high Hartmann number MHD duct flow with a cylinder placed near the heated side-wall. *International Journal of Heat and Mass Transfer* (2015) **67**:944–954.

[24] Baytaş C, Erdem D, Acar H. Okşan Ç.Y., 2008-2011. An experimental and investigation in the composite channel for the boundary conditions at a fluid- solid interface (2011).

[25] Ling CM, Jin YY, Z.Q C. Heat/Mass Transfer and Pressure Drop in a Triangular-rib-roughened Rectangular Channel, Int. *Journal of Heat and Fluid Flow* (1994) **15**:6–486.

[26] Huq M, Aziz-ul Huq AM. Experimental Measurements of Heat Transfer in An Internally Finned Tube. *International Communications in Heat and Mass Transfer* (1998) **25**(5):619–630.