EXPERIMENTAL INVESTIGATION AND FLOW STRUCTURE ANALYSIS OF DELTA WING

Gaurav Tote
Department of mechanical engineering, Sihngad Academy of engineering, Pune, India

Dr.Sandeep S. Kore
Department of mechanical engineering, Sihngad Academy of engineering, Pune, India

ABSTRACT

In this study the increase of heat transfer in a rectangular channel with triangular delta wing vortex generators is evaluated. These vortex generators can be mounted on the fin surfaces by either welding, punching or embossing. These vortex generators introduce stream wise longitudinal vortices. These vortices disrupted the growth of the thermal boundary layer and serves to bring about heat transfer augmentation between the fluid and the fin surfaces. Air is taken as the working fluid. The flow system is supposed to be turbulent because, usually the fin spacing is small and the mean velocity is such that the Reynolds numbers of interest are below the critical Reynolds number. The constant heat flux boundary condition is used.

This investigation work gives a performance data for a triangular wing in a plate-fin heat exchanger. In order to evaluate the performance, bulk temperature and average Nusselt number are calculated. The heat transfer enhancement is observed with the use of vortex generator. Also the velocity vectors and pressure counters in the test section around the different delta wings are analysed in CFD. Four different shapes of delta wings are evaluated with constant attack angle of 45^0 and constant pitch of 18mm between deltas. The study completed with different Reynolds number ranging from 8000 to 18000.

INTRODUCTION

Increasing demands on the performance of heat exchangers used in automotive industry, power systems, , electric circuit in electronic chip cooling, air conditioning and refrigerant applications, ,internal cooling of gas turbine blades and aerospace industry for reasons of compactness, reducing manufacturing cost and higher efficiency lead to use of heat transfer enhancement techniques. Vortex generation has emerged as one promising technique for enhancing air-side convection. In this process, wing like vortex generators (VGs) are punched or mounted on a heat-transfer surface to produce longitudinal vortices. There are two distinct methods for heat exchange augmentation:1) active vortex method and 2) passive vortex method. The active vortex method is used to actively govern the secondary flow and pressure drop so as to meet the required heat transfer rates even at the cost of increased pumping power. There is little use of this method in heat exchangers as the operating cost is high. A few case of active vortex method are the use of jets at different angles from the heat transfer surface into the boundary layer, and the production of a secondary flow through acoustic excitation. Using longitudinal or latitudinal vortex generators for heat exchange augmentation is known as the passive vortex method. Delta wing, rectangular wing, delta winglet, trapezoidal delta winglet, rectangular wing, dimpled surfaces, ribs, and fins all are types of vortex generators. Vortex generator is a kind of passive heat transfer enhancing device which is attached to the duct walls or fin surfaces and project into the flow at an angle of attack to the flow direction. It can be stamped on or punched out from the fin. Using VGs, the fluid flow can be forcefully disturbed because of the production of vortex when fluid flows over it. The vortex generator not only disturbs the flow field, disrupts the growth of the boundary layer, but also makes fluid swirl and causes a heavy exchange of core and wall fluid, guiding to the augmentation of heat transfer. The basic principle of vortex generators (VGs) is to create secondary flow, particularly longitudinal vortices, which disturb the thermal boundary layer developed along the wall and remove the heat from the wall to the core of the flow by means of high-scale turbulence.[1]

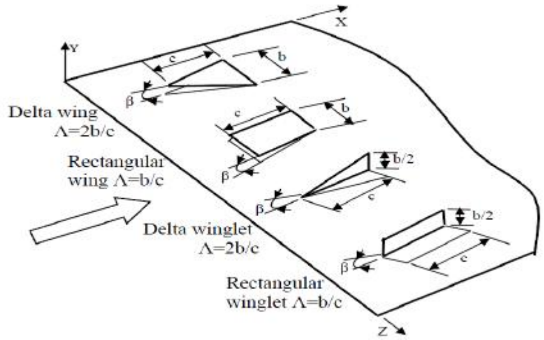


Fig no 1 Longitudinal vortex generators[1]

Enhancement of heat transfer through different means has been an strained area of research for many years. There are numerous applications where high performance heat exchange is desired. This research involves the numerical analysis of heat exchange augmentation in a rectangular channel using distinct types of longitudinal vortex generators (LVG) for a turbulent flow. A computational fluid dynamics software package was used to compute the 3-D steady viscous flows with heat transfer. The effects of Reynolds number ranging from 8000 to 20000 (turbulent flow) are shown. Four different geometries of delta wings are studied. The Nusselt number (Nu) is computed and compared with the Nusselt number (Nu) without the LVG's. The results show that the LVG's effectively enhances the heat transfer in the rectangular channel. In addition, the impact of the LVG's drag and the resulting pressure drop across the channel was quantified**.** The Darcy's friction factor (f) was computed and compared with the friction factor without LVG's (f_0). For each case the performance evaluation parameter was computed to gauge the overall efficiency of the configuration.

LITERATURE REVIEW

This study Inspired by bunch movement of animals in nature, a newly vortex generator (VG) array deployed in a "V" shape is proposed in this study, intention to create constructive interference between vortices and better VG performance. Its impact von surface convection enhancement is experimentally assessed in a development channel flow. A large-aspect-ratio duct of 6 mm high is constructed to model a single passage of plain-fin heat exchangers. The frontlet air velocity ranges from 0.9 to 2.5 m/s, corresponding to a Reynolds number range based on channel height of 340 to 940. The proposed V-array show superiority to a conventional multi-row configuration in that it affects a much larger heat transfer area, and the boost effect by the trailing pair is manifest even at relatively small Reynolds numbers. A two-pair V-array deployed at 30º yields 12-36% augmentation in the total heat transfer for the current channel flow, and is considered an appropriate design for implementation in prototype heat exchangers.[2]

Fig no 2 Migrating birds in V-formation[2]

An innovative design of triangular shaped secondary fins with rectangular or a delta wing vortex generator mounted on their slant surfaces for enhancing the heat transfer rate in plate-fin heat exchanger is proposed. The present analysis uses a moderate version of Marker and Cell method to solve the governing equations. The solution of Navier-Stokes equations gives clearly a provisional value of the velocity components to be used for the next time step. However, these clearly advanced velocity components may not yield a realistic flow field. Therefore, the continuity of flow is checked using these velocity components. The flow structure is visualized by the cross-stream velocity vectors along and downstream the wing and the results clearly depicted the generation of the vortices. In the case of stamped wing, some of the fluid gets entrained through the hole beneath the wing; thereby reducing the strength of the cross stream velocity vectors. To this end the results of the computation are expressed in terms of the compactness achieved by using the proposed design and about 32% reduction in length is possible by the use of delta wing vortex generator at an angle of attack of 26º. [3]

In this study the enhancement of heat transfer in a rectangular channel with triangular vortex generators is done. The span wise averaged Nusselt number, total heat flux and mean temperature are compared with and without vortex generators in the channel at a blade angle of 30° for Reynolds numbers 800, 1200, 1600, and 2000. The use of vortex generators increases the span wise averaged Nusselt number compared to the case without vortex generators. At a particular blade angle, increasing the Reynolds number results in an enhancement in the overall performance efficiency and span wise averaged Nusselt number was found to be greater at particular location for larger Reynolds number. The total heat flux from the bottom wall with vortex generators was found to be greater than that without vortex generators and the difference increases with increase in Reynolds number.[4]

In this study, an array of delta-wing vortex generators is applied to a plain fin-and-tube heat exchanger with a fin spacing of 8.5 mm. Heat transfer and pressure drop performance are measured to determine the effectiveness of the vortex generator under frosting conditions. For Reynolds numbers between 500 and 1200, a reduction of 35.0% to 42.1% is observed in the air-side thermal resistance. Correspondingly, the heat transfer coefficient is observed to be between 33-53 W/m2-K for the enhanced heat exchanger and between 18-26 W/m2-K for the baseline heat exchanger. A modified volume goodness parameter is also calculated and shows that the enhanced exchanger outperforms the baseline specimen for the range of Reynolds numbers examined.[5]

EXPERIMENTAL SETUP:

The test apparatus is an open air flow loop type Consist of centrifugal blower, flow control valve, orifice meter (for flow measurement), an entry section, the test section and plenum (exit section). The duct is of size 700mm * 50mm* 50mm and is constructed from epoxy resin of 5mm thickness. The test section is of length 700mm. The entry and exit section length are 240mm and 170mm respectively and is made up of 5mm thick acrylic sheet. The exit section of 170 mm is used after the test section in order to reduce the end effect, and to get uniform temperature across the duct. A 240 mm acrylic entrance section provides hydro-dynamically fully developed flow at the test section entrance. To connect rectangular cross section to circular cross section plenum chamber is used, in between exit section and orifice meter.

To provide uniform heat flux electric heater of size 700mm*50mm maintained wattage of 125 Watt is fabricated from galvanizing iron sheet of 0.5mm thickness of 1 amp and 230 volts. The heater is located at the bottom of plate and is covered by the asbestos sheet of size 700mm*50mmm*5mm to avoid the heat loss from bottom side of heater. Heat input to the heater is controlled by the use of dimmer range of 0 to 230 volts. The voltage across heater is measured by digital voltmeter. The whole periphery of the test section is covered by Ceramic wool layer of 6cm thickness, in order to minimize the heat loss to the surrounding by radiation. The mass flow rate of air is measured by means of an orifice meter and the flow is controlled by the gate valve provided in line. Pressure drop across the orifice meter was measured by a U- tube manometer with water as a manometer fluid. The pressure drop across test section is measured by Differential U- tube manometer, with double reservoir filled with benzyl alcohol and water.

For temperature measurement K type cl/al thermocouple are used at inlet and outlet and the test plate. Out of 12 thermocouples one thermocouple measure the inlet temperature, three used for exit temperature measurement and six thermocouples are embedded in test plate through 2mm drilled hole of depth 5mm from side wall of plate through 2mm drilled hole of depth 5mm from side wall of plate. Among six, three thermocouples embedded in plate from one side and remaining from other side of the test plate to measure average surface temperature. Four thermocouples are used at inlet and outlet of test duct to measure air temperature. A digital mili-voltmeter is used for measurement of thermocouple output through the selector switch

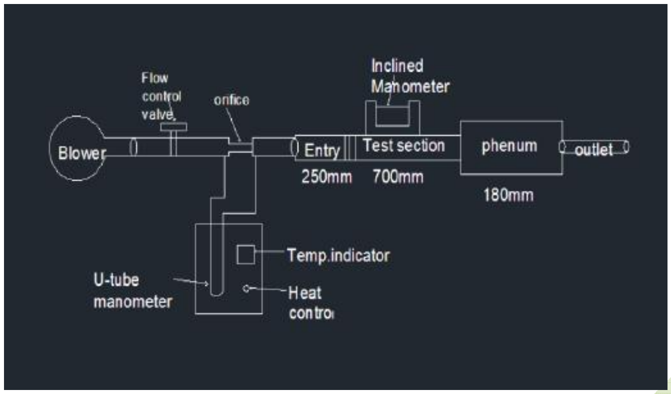


Fig no 3 Diagram of test rig

Experimental Procedure:

The test section is assembled in test bracket and checked for air leakage. The blower was switched on to let a Pre determined rate of air flow through the duct. First a Experimental validation process is carried out on flat plate. A constant heat flux is applied to the flat surface. The net heat flux and the average test surface to bulk mean air temperature difference was determined over test section. To collect relevant heat transfer and flow friction data test runs were conducted under steady state condition which is assumed to be reached when the temperature at a point does not change for about 15 minutes. After starting it from cold it took around 45 to 60 minutes to gain the system steady state. Six valves of flow rates were used for each set at same or fixed uniform heat uniform heat flux. At each valve of flow rate and the corresponding heat flux, system was allowed to attain a steady state before the temperature data were recorded. The pressure drop in test section is measured in cold state when heater is off.

After experimentation on test rig, the results are co-relate with the theoretical factors also the same study done numerically in CFD . The simulation is plotted in the form of images which provides the actual data of velocity counters and pressure drop within the test section.

CFD MODELING:
Geometry:

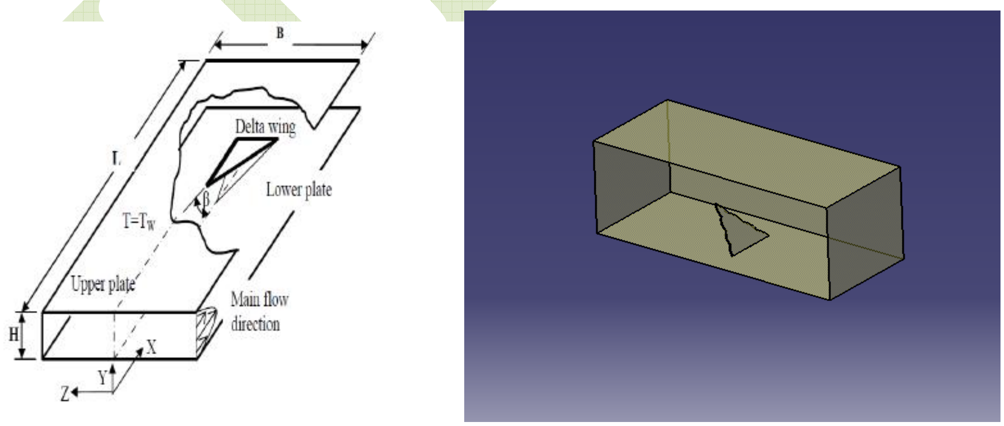


Fig no 4 Domain of Test section

Grid Generation:

In this study ANSYS Workbench (ICEM CFD) was used as the meshing tool. A structured non uniform grid was produce. Fig. shows the isometric view of the fluid domain meshing whereas finished meshing near the vortex generators is pictured in Fig 5. Fine meshing is required near the wall region to capture the boundary layer on the wall. The whole fluid domain has all the elements as tetrahedral elements also on delta surface prism layers are produce to better capture of geometry. Mapped Face Meshing option from the ANSYS Workbench meshing tool was used to generate this kind of structured non uniform meshing .the part setup was done as below for the each geometry.[5]

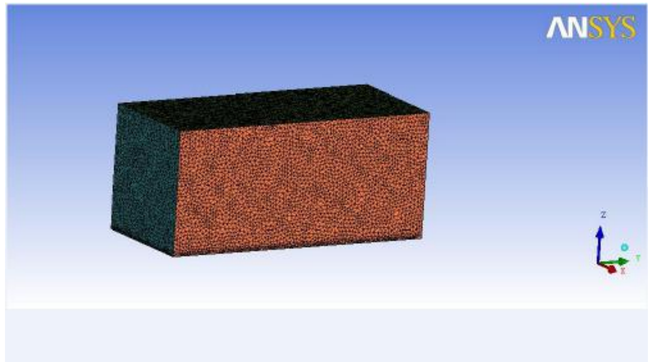


Fig no5 Meshing of Domain

Fluent Setup:

FLUENT 14.5 was used for CFD analysis in this study. The mesh files created in ANSYS Workbench called ICEM CFD and imported in fluent. The Fluent model is setup to allow energy equation in a Realizable *k*-Ű model with enhanced wall treatment. The fluid which is used in this study is air with a constant density of 1.225 kg/m3, dynamic viscosity of 1.7894 * 10-5 kg/m.s, the constant pressure specific heat of 4182 J/Kg-K, and thermal conductivity of 0.6 W/m.K. The operating condition on the interior of the channel is fluid. The boundary conditions are tabulated in Table I are applied on the rectangular channel including the inlet and outlet channel. A temperature of 358K is applied on the bottom surface of the rectangular channel. The inlet has been given an inlet temperature of 300 K and a specific velocity based on the Reynolds number corresponding to the chosen Reynolds number. The outlet was set at ambient condition. The walls of the whole channel, as well as surfaces of the vortex generator, have been given the no slip boundary condition. A second order upwind discretization method has been used for energy and momentum. Convergence is based on the absolute criteria of continuity, *x* velocity, *y* velocity and *z* velocity being equal to 10-3 and energy equal to 10-6. This means that the solution will converge once the residuals reach the above mentioned mark. The model is computed from the inlet surface and 500 iterations were given for the solution to converge. The flow is with turbulent intensity of 3 % and the velocity of the flow is calculated from [5]
The flow is turbulent and the velocity of the flow is calculated as,

$$\mathrm{Re} = \frac{\rho V H_d}{\mu},$$

Where, Re = Reynolds number, (dimensionless)
ρ = density of the fluid, (kg/m3)
V = mean velocity of fluid flow, (m/s)
Hd = characteristic length or hydraulic diameter, (m)
μ = dynamic viscosity of the fluid, (kg / (m·s))

The hydraulic diameter is taken as

$$H_d = \frac{4 * \text{Area of the rectangular cross section}}{\text{Perimeter of the rectangular cross section}}$$

$$H_d = \frac{4 * B * H}{2(B + H)}$$

The heat transfer coefficient is used to calculate the convective heat transfer

$$h = \frac{Q}{A \cdot \Delta T}$$

where, Q = heat transfer capacity, (W)

A = heat transfer surface area, (m2)
ΔT = log mean temperature difference, (K)

Table I BOUNDARY CONDITIONS

Zone	**Assigned Boundary Type**
Inlet	Velocity Inlet, Velocity of air = 5.5 m/s Inlet fluid temperature = 300K
Outlet	Pressure Outlet, 0 Pa gauge
Bottom wall	Wall (No Slip), Constant Temperature = 358 K
Top wall	Wall (No Slip), Constant Heat Flux = 0 W/m2
Side walls	Wall (No Slip), Constant Heat Flux = 0 W/m2
Vortex Generator	Wall (No slip), Constant Heat Flux = 0 W/m2

RESULT AND DISCUSSION

During Experimentation the following parameters were measured:
1. Pressure difference across the orifice meter.
2. Temperature of the heated surface and temperature of air at inlet and outlet of the test section.
3. Pressure drop across the test section using differential manometer.
4. Voltage and current value for constant heat flux.

Validation of Experimental Setup

Table 2 Temperature difference at various Re no

Re	**T_s (ºC)**	**T_{ai} (ºC)**	**T_{ae} (ºC)**
8000	81.26	30.05	33.22
10000	79.83	30.08	33.07
12000	77.86	30.19	32.84
14000	73.65	30.28	32.74
16000	70.56	30.32	32.56
18000	69.67	30.08	32.18

The thermo physical properties of air used in calculation of heat transfer are taken from air table corresponding to bulk mean temperature. The Dittus Boelter equation for Nusselt number is valid for the Reynolds number range above 5000.

$$Nu_{th.} = 0.023\, Re^{0.8}\, Pr^{0.4}$$

Table 3 Thermo physical properties of air at various Re no

Re	**ΔT**	**Tbm**	**Ts - Tbm**	**Q**	**h**	**Nu0**	**Nu_{expt}**	**% Deviation (+/-)**
8000	3.17	31.64	49.62	26.17	15.07	28.06	28.61	9.70
10000	2.99	31.58	48.26	28.50	16.88	31.43	32.10	4.24
12000	2.65	31.52	46.34	30.94	19.08	35.54	37.75	6.34
14000	2.46	31.51	42.14	33.16	22.48	41.89	42.35	3.34
16000	2.24	31.44	39.12	33.76	24.66	45.95	46.31	2.02
18000	2.10	31.13	38.54	36.09	26.76	49.91	51.44	1.37

Table 4 Comparison of theoretical and experimental Nu

Re no	Nu_{expt}	Nu_{th}
8000	28.61	28.06
10000	32.10	31.43
12000	37.75	35.54
14000	42.35	41.89
16000	46.31	45.95
18000	51.44	49.91

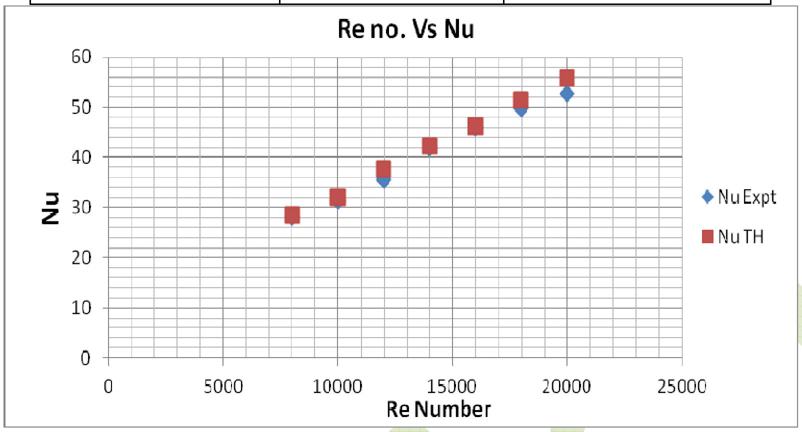
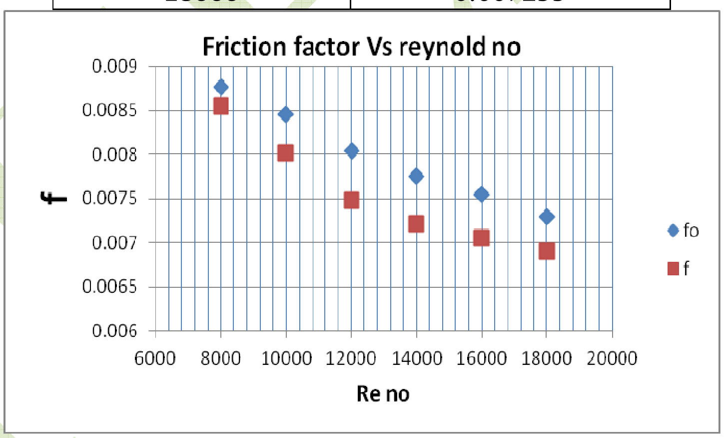
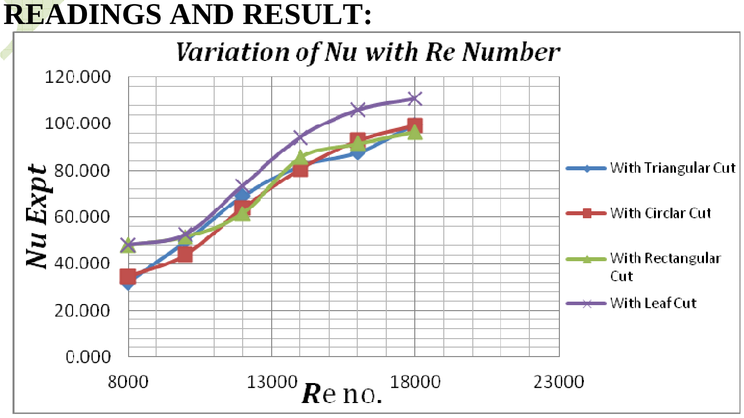


Table 5 Friction factor at various Re no

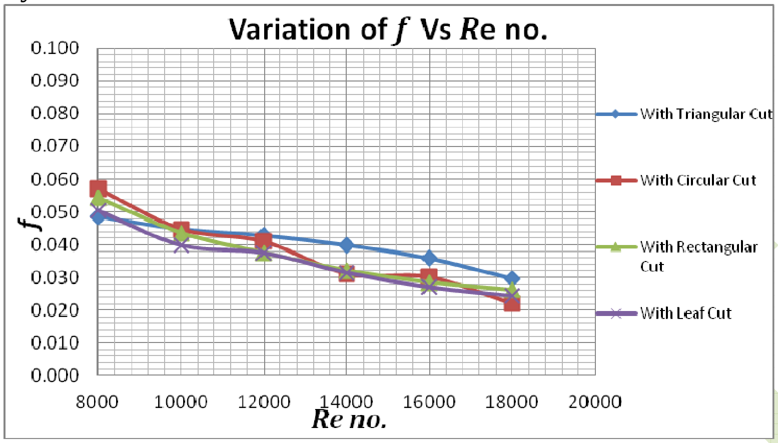
Re no	fo
10000	0.008458
12000	0.00804
14000	0.007756
16000	0.007543
18000	0.007299



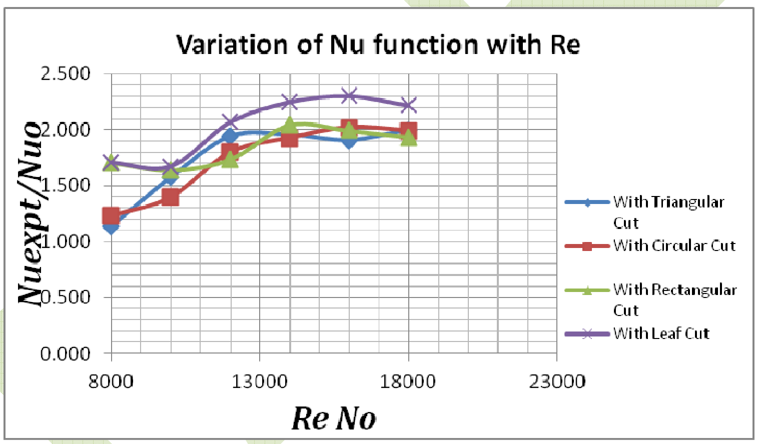
EXPERIMENTAL READINGS AND RESULT:



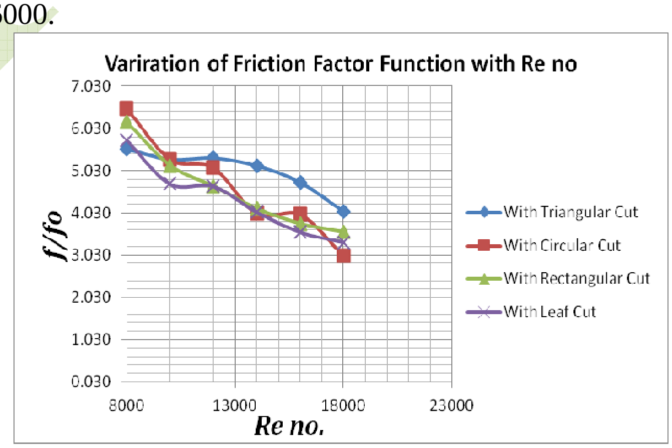
In above graph, The experimental nusselt number is plotted against Reynolds number and it is observed that, 1] A Experimental nusselt number is increasing with the increase in Reynolds number. 2] Out of four different shapes of delta, a leaf cut delta having maximum nusselt number for all Reynolds number and it is ranging between 48 to 112. 3] Among remaining three shapes, all have nearly same variations with all Reynolds number.



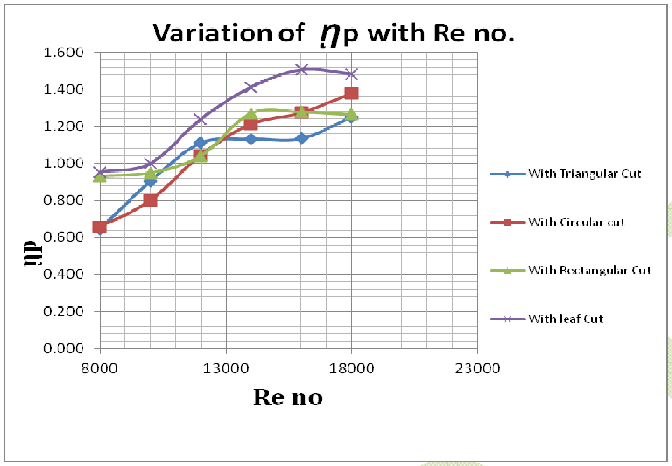
In above graph, The variation of friction factor is plotted against Reynolds number. It is observed that, 1] the friction factor gets reduced with increase in Reynolds number. 2] The circular cut deltas has maximum drop in friction factor up to 0.026. 3] The friction factor is ranging within 0.024 to 0.056 for all geometries. 4] The triangular cut geometry has minimum range of friction. it varies from 0.048 to 0.03.



In above graph , The variation of Nusselt no factor is plotted against Reynolds number and It is observed that , 1] The leaf cut delta geometry has maximum factors for all ranges of Reynolds number. The maximum factor is 2.3 at 16000 Re. This is happened due to less and smooth cuts on edges.
2] For other three geometries, the factor ranges from 1.1 to 2 . Particularly a rectangular cut has uneven changes in nusselt no factor. 3] it is observed that, all the geometries have decreasing nature at higher Reynolds no i.e after 16000.

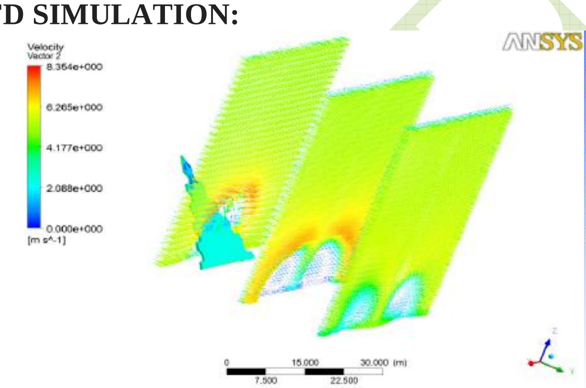


In above graph , The variation of friction factor ratio is plotted against Reynolds number and It is observed that , 1] The triangular cut has a minimum range of ratio and it varies from 5.050 to 4.030. 2] The highest range of friction factor ratio is achieved by circular cut and it varies from 6.090 to 3.030. 3] The friction factor ratio mostly goes on increasing but due to bigger geometry of delta wings, friction factor is high.



In above graph, The variation of friction factor ratio is plotted against Reynolds number and It is observed that, 1] The leaf cut geometry has maximum performance efficiency 0f 1.5 times because the geometry has minimum curvature and has aerodynamic shape. 2] The second highest efficiency is achieved by circular cut geometry up to 1.4 times Because circular cuts did not produce that much friction on the edges so it increase efficiency. 3] The triangular and rectangular cut geometry has minimum efficiency of 1.22 times.

CFD SIMULATION:



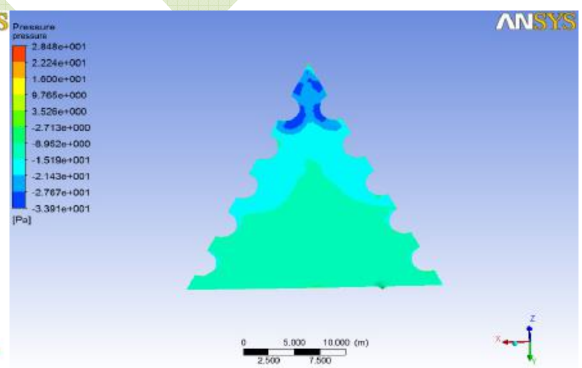


Fig no 6 Velocity Vectors of circular cut wing at 5.5 m/s air velocity *Fig no7 Pressure contours on circular wing at 5.5 m/s air velocity*

CONCLUSION

An experimental study of the flow of air in a rectangular channel with Delta wings on flat plate, subjected to uniform heat flux boundary condition has been performed. The effect of Reynolds number and delta wing geometries on the heat transfer coefficient and friction factor has been studied. Experimental results measured on delta wing geometries surfaces placed on flat plate in channels with an aspect ratio of 2 are given for Reynolds numbers from 8,000 to 18,000. Four different geometries i.e Triangular cut, rectangular cut, Circular cut, and leaf cut are tested in this study. The pitch is constant for all geometry. The experimental results have been compared with those of smooth flow channel under similar flow conditions to determine enhancement in heat transfer coefficient and friction factor.

The following conclusions are made,

1] Heat transfer improvement with delta wings seems to have a maximum value of approximately 1.5 and overall maximum thermal performance of about 1.5 for a depth leaf cut geometry.

2] The second highest efficiency is achieved by circular cut geometry up to 1.4 times Because circular cuts did not produce that much friction on the edges so it increase efficiency.

3] Other two geometries that is triangular cut and rectangular cut has nearly equal performance efficiency that is 2.024.
4] In CFD simulation, It is observed that, The negative pressure was created on the upper surface of delta near the tip for all geometry.
5] The turbulent intensity goes on increasing as Reynolds no increases.
6] The velocity vectors are generated behind the vortex generator having distinct nature for different Reynolds number.

REFERENCES

[1] NINAD C. MANIAR (2012)”Heat Transfer Enhancement In A Rectangular Channel Using Vortex Generator In A Laminar Flow” The University Of Texas At Arlington, December 2012
[2] He, Jing and Jacobi, Anthony M., "Air-Side Heat Transfer Enhancement by a V-Formation Delta-Winglet Array in a Developing Channel Flow" (2010). *International Refrigeration and Air Conditioning Conference* Paper 1140
[3] Sachdeva, G**.**, Vasudevan, R., and Kasana, K. S., 2009, “Computation of Heat Transfer Enhancement in a Plate-Fin Heat Exchanger with Triangular Inserts and Delta Wing Vortex Generator,” *International Journal for Numerical Methods in Fluids*, Published On-Line, DOI 10.1002/fld 2113, Wiley Publications
[4] M. C. Gentry and A. M. Jacobi. (1995). Heat Transfer Enhancement on a Flat Plate
Using Delta-Wing Vortex Generators, *Prepared as part of ACRC Project 40 Vortex-Induced Air-Side Heat Transfer Enhancement in Air-Conditioning and Refrigeration Applications.*
[5] Sommers, Andrew and Jacobi, Anthony M., "Air-Side Heat Transfer Augmentation of a Refrigerator Evaporator Using Vortex Generation" (2004). *International Refrigeration and Air Conditioning Conference.* Paper 714
[6] N. K. Singh, Numerical Investigation of Heat Transfer in a Channel with Delta Winglet Vortex Generators at Different Reynolds Numbers, World Academy of Science, Engineering and Technology International Journal of Mechanical, Aerospace, Manufacturing, Industrial Science and Engineering Vol:7 No:12, 2013