RESEARCH ARTICLE OPEN ACCESS

Analysis of A Double Spiral Counter Flow Calorimeter in Impinging Flame Jet Using CFD Tool

Santosh Bagewadi, A M Mulla, Dilip Sutraway
(Assistant Professor Department of Mechanical Engineering SECAB I E T Vijaypur, Karnataka, India-586101)

ABSTRACT
Enhancement of heat transfer rates in heat exchanger and calorimeter has been reported by many researchers. However, work regarding heat transfer characteristics analysis of double spiral counter flow calorimeter is not published and this forms the objective of this work. DSCFC is a unique design where it consists of single fluid as working fluid for heat exchange. Here heat transfer takes place between solid and fluid, and hence can be called as conjugate heat transfer problem. Heat transfer characteristics DSCFC is observed at various Reynolds number and base temperature. DSCFC is analyzed considering conjugate heat transfer and temperature dependent properties of heat transport media. Computations are performed using commercially available CFD package ANSYS-CFX. It is observed that with increase in Reynolds number of the fluid, heat transfer reduces whereas increase in base temperature increases heat transfer. The Computational results are compared with the experimental.

Keywords: Double spiral calorimeter, Computational Fluid Dynamics, Conjugate heat transfer.

I. INTRODUCTION

A calorimeter is an object used for calorimetry, or the process of measuring the physical changes as well as heat capacity. The cooling or heating jacket controls the temperature of the process. Heat is measured by monitoring the heat gained or lost by the heat transfer fluid. Constant flux calorimetry is derived from heat balance calorimetry and uses specialized control mechanisms to maintain a constant heat flow (or flux) across the vessel wall.

Flame jet impingement heat transfer is a very important process in industry and is used for many applications like melting of scrap metal, shaping of glass, welding, etc. Other advantage is that by this method we can reduce the fuel consumption hence we can increase the efficiency of the system of the industry where heating is used. A lot of research work has been carried in this area, both experimental and numerical. Moreover, during practical analysis the target surface is curved (i.e. cylindrical or spherical). All experimental analysis where made by taking the water as the cooling media, on copper or brass plate and constant temperature boundary condition where applied.

Double Spiral Counter Flow Calorimeter (DSCFC) of constant flow area of 80 mm2 will be used and the heat transfer takes place from flame to calorimeter base plate/ target plate and then from constant temperature base plate (isothermal) to spiral flow water, it attains not only high heat transfer co-efficient but also improved cooling uniformity and maintain constant base plate temperature. In this project a double spiral counter flow calorimeter set-up is studied and it consists of spiral flow domain of two concentric channels as shown in Figure 1. In a DSCFC, the cold fluid enters inlet, flows inward and comes out outward as hot fluid by absorbing the energy from base plate.

Computational fluid dynamics is the branch of fluid dynamics providing a cost effective means of simulating real flows by the numerical solution of governing equation. As a result of these factors, Computational Fluid Dynamics is now an established industrial design tool, helping to reduce design timescales and improve processes throughout the engineering world. CFD provides a cost-effective and accurate alternative to scale model testing, with variations on the simulation being performed quickly, offering obvious advantages.

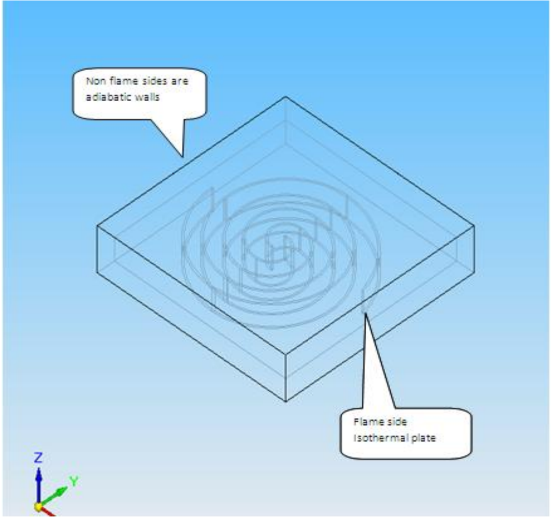


Fig.1 Double Spiral Counter Flow Calorimeter

II. Literature Review

Baukal et al [1] (1995) studied the targets were used cylinders, flat plates and hemi-nosed cylinders. Forced convection (laminar and turbulent) and thermo-chemical heat release, have been the most important heat transfer processes. Several semi-analytic solutions have been developed, for the heat flux to the forward stagnation point of a body of revolution. These were originally developed for aerospace applications, such as rocket re-entry into the earth's atmosphere. These solutions, and many variations, have been used to simulate flame impingement heat transfer. The results of sample calculations are compared to some of the experimental measurement. Twelve flame impingement experimental studies have been considered here. In those studies, the measured heat flux has been compared against one or more semi-analytic solutions. Cylindrical, flat plate, and hemisphere cylindrical targets have been used in one, three, and nine studies respectively.

Baukal et al [2] (1997) reported on Surface Condition Effects on Flame Impingement Heat Transfer, This study investigated the heat transfer from oxygen-enhanced, natural gas flames (15 kW) impinging normal to a water-cooled metal disk (d b = 135 mm) segmented into concentric calorimetric rings. Polished, untreated, and blackened surfaces were used to study emissivity effects. The heat flux to the blackened and polished surfaces was the highest and lowest, respectively. The flux to untreated surfaces was between the highest and lowest fluxes. The largest difference in the flux, between the polished and blackened surfaces, was only 9.8%. Catalytic effects were investigated by using alumina-coated (nearly non-catalytic), untreated, and platinum-coated (highly catalytic) surfaces. The heat flux to platinum-coated surfaces was the highest. The fluxes to untreated surfaces were similar to those for alumina-coated surfaces. The largest difference in the flux, between the platinum-coated and the alumina-coated surfaces, was only 12%. Therefore, both non luminous flame radiation and the thermo chemical heat release from surface catalytic reactions were relatively small fractions of the total heat flux. Inlet and outlet water temperature measured, from which heat equation were calculated.

Dong et al [3] (2003) studied the experimental investigation of the flame shape and the heat transfer and wall pressure characteristics of a pair of laminar premixed butane/air flame jets impinging vertically upon a horizontal water-cooled flat plate at jet Reynolds numbers of 800, 1000 and 1200, respectively. Equivalence ratio of the butane/air mixture was maintained constantly at unity. The flame shape, the pressure distribution on the impingement plate and the heat transfer from the flame to the plate were greatly influenced by the interference occurred between the two flame jets. This interference caused a sharp pressure peak at the between-jet midpoint and the positive pressures at the between-jet area, which led to the separation of the wall jet from the impingement plate after collision. The flame impingement surface was a square copper plate of 200 mm long, 200 mm wide and 8 mm thick. It was uniformly cooled on its backside by a cooling water jacket. Copper was selected to fabricate the plate because of its excellent thermal conductivity. The top plate of the cooling water jacket was made of Plexiglas to enable the water flow to be visualized. A stainless steel frame was used to support the copper plate and the cooling water jacket, so that the plate could be placed either horizontally or tilted at a selected angle relative to the burner. After a change in the operating condition had been made, measurements were only conducted after the steady-state conditions had been established.

Shuhn-Shyurng Hou et al [4] (2003) studied the measurements of temperature distributions, a water-cooled stainless plate, 20 mm thick and 100 mm in diameter, is used as the stagnation plane, as shown in Fig. 1. The cooling water, supplied by a circulatory (thermostatic water container), flows into the water-cooled plate and removes the heat from the flame. The thermal efficiency of an impinging flame is defined as the percentage of the thermal input transferred to the cooling water. Therefore, the thermal efficiency equation can be determined by measuring temperature difference between the inlet and outlet cooling water,

Subhash Chander et al [5] (2005) used copper plate of 8 mm thickness and 300 mm diameter as the impingement surface. The surface of the plate was smooth and it did not have any coating on it. There was no soot deposition on the surface; still as precaution it was periodically cleaned. A water jacket was provided at the rear of the copper plate to evenly cool the plate from the backside. Water flows into the calorimeter at the center and comes out from the calorimeter through the two exits provided at diametrically opposite points. Inlet and outlet temperatures of the water were measured with T type thermocouples with full-scale accuracy of ±0.5%.

Tuttle et al [6] (2004) fabricated impingement plate, measuring 71 cm square, with two layers of aluminum plate separated by a rubber seal. The bottom plate, made of 7075-T6 aluminum of thickness 1.27 cm is exposed to the impinging flame, with milled parallel grooves through which cooling water flows. The flow of the cooling water was regulated with three calibrated variable area flow meters. Thermocouples were used to measure the temperature of the water flowing in and out of the plate to maintain the plate at a near-constant temperature.

Kwok et al [7] (2004) studied three flame jets impinged vertically upward onto a water-cooled copper impingement plate having a surface area of 500 • 500 mm2 and a thickness of 8 mm. The cooling water was kept at a temperature of 38oC by a refrigerator to eliminate the condensation of water vapour on the impingement plate surface. A heat flux transducer having an effective sensing area of 6 mm2 was installed at the centre of the flame-side surface of the impingement plate to measure the local heat flux from the flame to the plate. By moving the three-dimensional positioner horizontally in the x–y plane, the local heat flux of a point on the impingement plate relative to the stagnation point could be measured, such that the heat flux distributions in the x- and y directions of the impingement plate were obtained. The heat flux received by the impingement plate was reduced when the nozzle-to-plate distance was increased.

Van Der Meer et al [8] (1991) studied the two thermo-stated baths provided water flow of a constant temperature. This flow was forced to pass through a channel between a copper plate and a glass plate separated by a small distance of 2.9 mm. The water flowed along the copper plate into a reservoir behind the plate and from that reservoir back into the first of the two thermo-stated baths. In this way one side of the glass plate was kept to a nearly constant temperature provided the flow in the channel was homogeneous and the heat transfer coefficient to the glass plate very high. The temperature of the water flowing through the channel was measured by thermocouples. The other side of the glass plate was covered with a thin layer of liquid crystals to measure its surface temperature.

Jayakumara et al [9] (1997) studied heat transfer characteristics of double pipe helical heat exchangers. Heat transfer characteristics inside a helical coil for various boundary conditions are compared. It is found that the specification of a constant temperature or constant heat flux boundary condition for an actual heat exchanger does not yield proper modeling. Hence, the heat exchanger is analyzed considering conjugate heat transfer and temperature dependent properties of heat transport media. Experimental results are compared with the CFD calculation results using the CFD package FLUENT 6.2. Based on the experimental results a correlation is developed to calculate the inner heat transfer coefficient of the helical coil.

Achmad Nursyamsu [10] (2007) analyzing the flow of water in a horizontal spiral pipe by comparing the pitch distance 236 mm, 246 mm, 256 mm in a spiral pipe with a smooth surface. This is to determine how far the effects of changes in pitch on pressure drop are obtained and also to determine the velocity vector of water flow in horizontal spiral pipe using CFD method. By making a model of spiral pipe and done meshing using hybrid simulation configuration element type or tetrahedron, with meshing the volume and size intervals of 5, and make the boundary condition using the value V = 0.73 m / s and Dh = 0032 m, so that pressure loss is obtained and velocity at 246 mm pitch with a smaller decrease in pressure (896.4 Pa) and the velocity distribution of larger (max: 1.70 m / s, min: 5.39 x 10-2 m / S).

III. The solid modeling of DSCFC in CFD-ICEM

Figure 2 shows the construction of spiral strip on copper base plate, this strip forms a spiral flow passage over the base plate. Figure-3 show the top plate constructed on spiral strip, where adiabatic boundary condition is applied

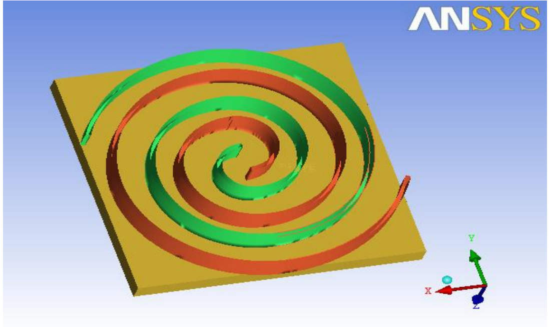


Fig-2 Base plate with spiral strip

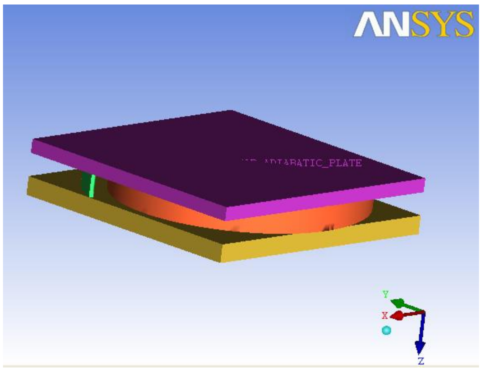


Fig-3 complete model

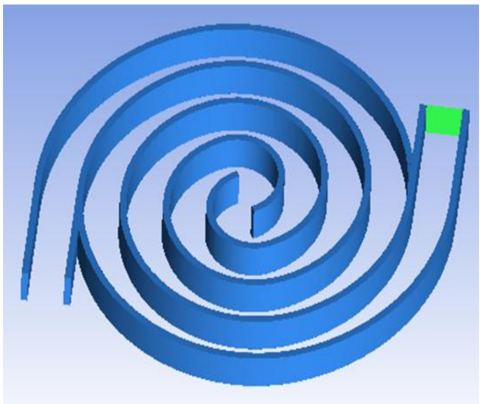


Fig-4 Inlet flow position

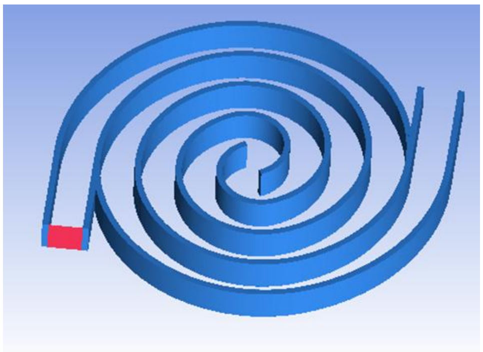


Fig-5 outlet flow position

Boundary and Initial Conditions:

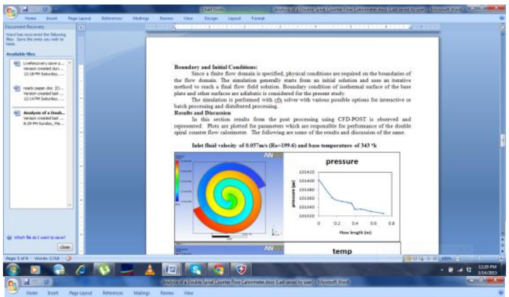
Since a finite flow domain is specified, physical conditions are required on the boundaries of the flow domain. The simulation generally starts from an initial solution and uses an iterative method to reach a final flow field solution. Boundary condition of isothermal surface of the base plate and other surfaces are adiabatic is considered for the present study.

The simulation is performed with cfx solver with various possible options for interactive or batch processing and distributed processing.

IV. Results and Discussion

In this section results from the post processing using CFD-POST is observed and represented. Plots are plotted for parameters which are responsible for performance of the double spiral counter flow calorimeter. The following are some of the results and discussion of the same.

For inlet fluid velocity of 0.057m/s (Re=199.6) and base temperature of 343 ok. The pressure, temperature and velocity profile are shown as figure-6



Boundary and Initial Conditions:

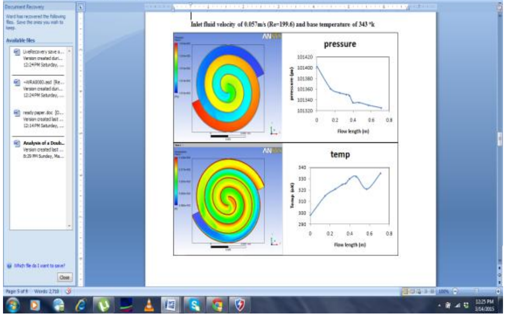
Since a finite flow domain is specified, physical conditions are required on the boundaries of the flow domain. The simulation generally starts from an initial solution and uses an iterative method to reach a final flow field solution. Boundary condition of isothermal surface of the base plate and other surfaces are adiabatic is considered for the present study.

The simulation is performed with cfx solver with various possible options for interactive or batch processing and distributed processing.

Results and Discussion

In this section results from the post processing using CFD-POST is observed and represented. Plots are plotted for parameters which are responsible for performance of the double spiral counter flow calorimeter. The following are some of the results and discussion of the same.

Inlet fluid velocity of 0.057m/s (Re=199.6) and base temperature of 343 °k



Inlet fluid velocity of 0.057m/s (Re=199.6) and base temperature of 343 °k

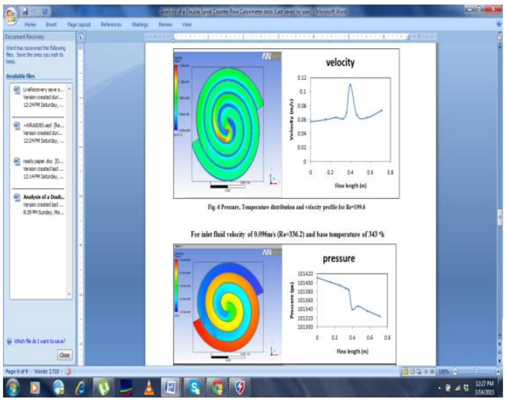


Fig-6 Pressure, Temperature distribution and velocity profile for Re=199.6

For inlet fluid velocity of 0.096m/s (Re=336.2) and base temperature of 343 °k

Fig-6 pressure, temp and velocity profile

For inlet fluid velocity of 0.096m/s (Re=336.2) and base temperature of 343o k. The pressure, temperature and velocity profile are shown in figure-7

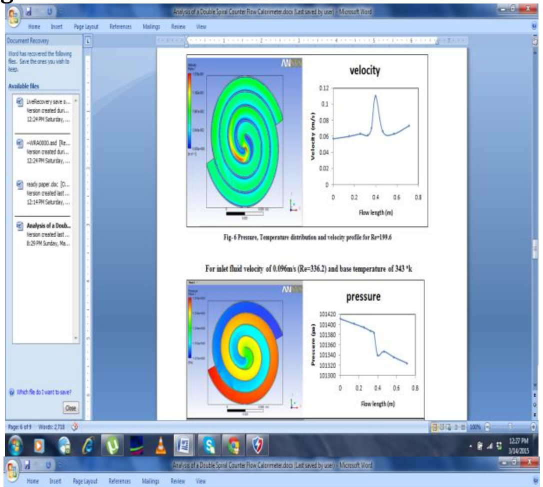
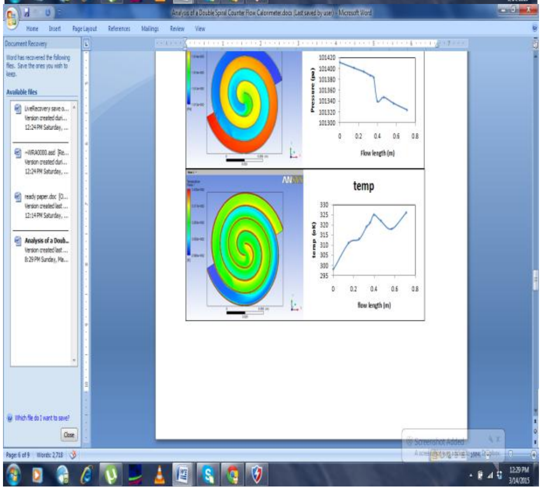
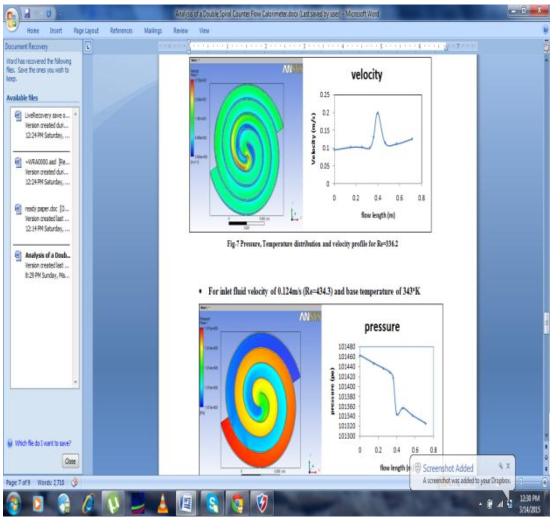


Fig-6 Pressure, Temperature distribution and velocity profile for Re=199.6

For inlet fluid velocity of 0.096m/s (Re=336.2) and base temperature of 343 °k





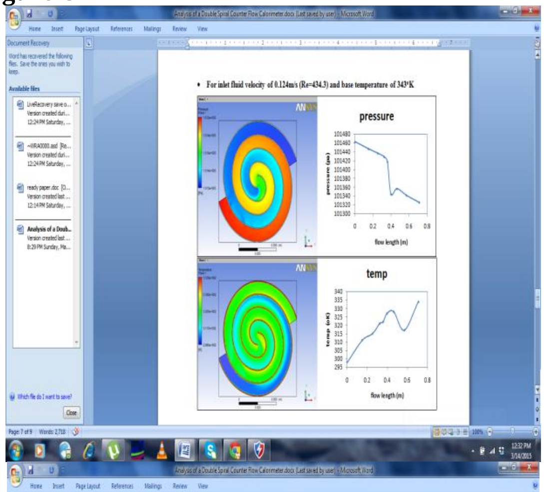
Word has recovered the following files. Save the ones you wish to keep.

Fig-7 Pressure, Temperature distribution and velocity profile for Re=336.2

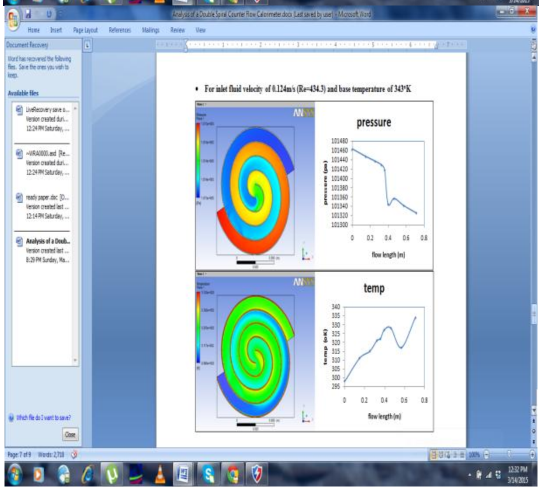
- For inlet fluid velocity of 0.124m/s (Re=434.3) and base temperature of 343°K

Fig-7 pressure, temprature and velocity profile.

For inlet fluid velocity of 0.124m/s (Re=434.3) and base temperature of 343o K. The pressure, temperature and velocity profile as shown in figure-8



- For inlet fluid velocity of 0.124m/s (Re=434.3) and base temperature of 343°K



- For inlet fluid velocity of 0.124m/s (Re=434.3) and base temperature of 343°K

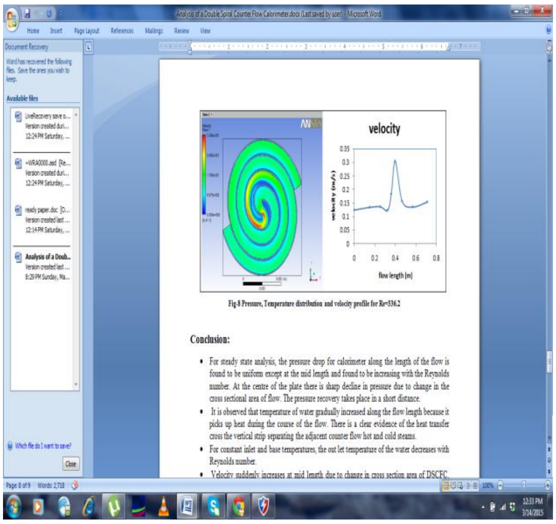


Fig-8 Pressure, Temperature distribution and velocity profile for Re=336.2

Conclusion:

- For steady state analysis, the pressure drop for calorimeter along the length of the flow is found to be uniform except at the mid length and found to be increasing with the Reynolds number. At the centre of the plate there is sharp decline in pressure due to change in the cross sectional area of flow. The pressure recovery takes place in a short distance.
- It is observed that temperature of water gradually increased along the flow length because it picks up heat during the course of the flow. There is a clear evidence of the heat transfer cross the vertical strip separating the adjacent counter flow hot and cold steams.
- For constant inlet and base temperatures, the out let temperature of the water decreases with Reynolds number.
- Velocity suddenly increases at mid length due to change in cross section area of DSCFC,

Fig-8 pressure, temprature and velocity profile.

V. CONCLUSION:

For steady state analysis, the pressure drop for calorimeter along the length of the flow is found to be uniform except at the mid length and found to be increasing with the Reynolds number. At the centre of the plate there is sharp decline in pressure due to change in the cross sectional area of flow. The pressure recovery takes place in a short distance.

It is observed that temperature of water gradually increased along the flow length because it picks up heat during the course of the flow. There is a clear evidence of the heat transfer cross the vertical strip separating the adjacent counter flow hot and cold steams.

For constant inlet and base temperatures, the out let temperature of the water decreases with Reynolds number.

Velocity suddenly increases at mid length due to change in cross section area of DSCFC, velocity recovers in a short distance in the downstream and remains almost constant till the outlet.

There is no evidence of the influence of the base temperature on the velocity distribution.

REFERENCES

[1] Baukal C.E. and Gebhart B. (1995) "*A review of semi-analytical solutions for flame impingement heat transfer*" Department of Mechanical Engineering and Applied Mechanics, University of Pennsylvania, Philadelphia, PA 19104-6315, U.S.A

[2] Baukal C.E. and Gebhart B. (1997) "*Surface Condition Effects on Flame Impingement Heat Transfer*" Mechanical Engineering and Applied Mechanics, University of Pennsylvania, Philadelphia, Pennsylvania

[3] Dong L.L, Leung C.W and Cheung C.S. (2003) "*Heat transfer and wall pressure*

characteristics of a twin premixed butane/air flame jets“ Department of Mechanical Engineering, The Hong Kong Polytechnic University.

[4] Shuhn-Shyurng Hou ,Yung-Chang Ko (2003) “*Effects of heating height on flame appearance, temperature field and efficiency of an impinging laminar jet flame used in domestic gas stoves*” Department of Mechanical Engineering, Kun Shan University of Technology, Tainan

[5] Subhash Chander, Anjan Ray (2005) “*Heat transfer characteristics of three interacting methane/air flame jets impinging on a flat surface*” Department of Mechanical Engineering, Indian Institute of Technology, Delhi, New Delhi

[6] Tuttle S.G., Webb B.W and McQuay M.Q. (2004) “*Convective heat transfer from a partially remixed impinging flame jet* “ Department of Mechanical Engineering, Brigham Young University, Provo, United States Received

[7] Kwok L.C., Leung C.W, Cheung C.S. (2004) “*Heat transfer characteristics of an array of impinging pre-mixed slot flame jets*” Department of Mechanical Engineering, The Hong Kong Polytechnic University,

[8] Van Der Meer T. H. (1991) “*Stagnation Point Heat Transfer from Turbulent Low Reynolds Number Jets and Flame Jets*” Faculty of Applied Physics, Delft University of Technology, Delft, The Netherlands

[9] Jayakumara J.S, Mahajania S.M., Mandala J.C, Vijayanb P.K. (1997) “*Experimental and CFD estimation of heat transfer in helically coiled heat exchangers*”

[10] Achmad Nursyamsu (2007). “*Analysis of flow mal-distribution in tubular heat exchangers by fluent”* Department of Mechanical Engineering National Institute of Technology Rourkela.