Numerical Simulation of Fluid Flow in Annuli with Groove in Inner Pipe for Different Reynolds Number and Aspect Ratios

Shailesh K. Patel, Research Scholar, FE&T, Ganpat University, Ganpat Vidhyanagar-384012, India, skpatel.mech@gmail.com

Dilip S. Patel, Research Scholar, FE&T, Ganpat University, Ganpat Vidhyanagar-384012, India, pateldilips@gmail.com

Dr. Kedar A. Pathak, Associate Professor, School of Science & Engineering, Navrachana University, Vadodara, India , kedarapathak@gmail.com

Abstract - **The paper presents results of numerical investigation carried out to study the effect of groove on inner pipe of double pipe heat exchanger (DPHE). . The objective of this work is to view the vortices generated in the cavity formed in groove and the flow pattern, pressure and velocity contours for further augmentation of heat transfer enhancement. The rectangular grooves were incised on outer surface of inner tube side with circumferential pattern and two different grooves depth, namely 0.3, 0.5 and 1 mm. The distance between grooves and the grooves high were kept constant, 8 mm and 0.3- 1 mm respectively. The tube diameter is 32 mm and it's made of copper. The shell is made of acrylic which has 82 mm inside diameter. Air is used as working fluid in annular passage. The smaller grooves space and lesser groove depth has more advantage since the recirculating region are increased which essentially cause larger heat transfer enhancement. The turbulence effect generated by groove near tube wall is responsible for increase of local temperature and finally bulk mean temperature and the heat transfer coefficient will increase. Thermal performance and high-pressure drop is an increasing function of groove depth, width and Reynolds number. Understanding the effect of turbulence on optimal heat transfer parameter and its further use for design of double pipe heat exchanger (DPHE) will be the outcome of this paper.**

Keywords —Grooves, *Heat transfer Augmentation, Turbulence promoter, Concentric annuli*

I. Introduction

A heat exchanger is a device for heat transfer that exchanges the heat at different temperature between working fluids [1]. One of the modestly heat exchanger is double pipe heat exchanger (DPHE). This kind of heat exchanger has many advantages such as serviceable easily, wide range temperature and pressure application. The thermal performance of the double pipe heat exchanger is the temperature driving force which can be measured from the Logarithmic Mean Temperature Difference (LMTD). One of the main conditions which affect the thermal performance is fluid flow. The problem of heat transfer through the boundary layer of the fluid destructively affects many fields in engineering such as chemical, transportation, combustion, cooling system and many others field applications. Special attention has been given to small heat exchanger but efficient in heat transfer. There have been continuous attempts to increase the heat transfer rate. The heat transfer between the inner and outer tube of heat exchanger can be improved by either increasing the heat transfer surface area using extended and corrugated surfaces without enhancing heat transfer coefficient or by increasing heat transfer coefficient using the turbulence promoters inside the tube. Different active and passive methods were developed in last few years. Active method uses external power source to enhance the heat transfer like surface vibration, and makes it suitable for limited application. There are many efforts in flow and heat transfer improvement through the passive method for today's technology [2,8]. Especially in passive method like twisted tapes, wire coils, helical ribs, fins, dimples, etc., surface modification has been developed for increasing the heat transfer and pressure drop in turbulent flow regime, this method adopting surface techniques that induced vorticities at the secondary flow region. Grooved heat exchanger would be the candidate for these needed. Grooves increase the surface areas, easy to install, little in space and weight. [3-6] investigated grooved pipe flow characteristics and their correlation on pressure drop and friction factor. To prove the phenomena behind the flow, this study completes with the simple visualization analysis. The grooves eliminate the flow oscillation and prevent the horse shoes vortex formation [7]. In this condition, the grooves were able to stop the turbulence spot formation and

 DOI : 10.18231/2454-9150.2018.0198

ISSN : 2454-9150 Vol-04, Issue-02, May 2018

lead to a decay of perturbation. Research has been done on various shape of groove in order to controlling fluid flow. [1, 2] investigated the approach temperature of grooved double pipe heat exchanger and the influence of groove toward heat transfer enhancement.
The objective of this study is to understand fluid flow behavior and visualize the vortexes generated numerically. We put attempt to understand the effect of groove depth on turbulence i.e. pressure-velocity contours give an effect to the heat transfer and pressure drop. The objective of this work is to visualize the enhanced heat transfer performance by generating swirl flow and vortex with grooves height. This study concentrates on observing fluid flow behavior through the grooved annulus in double pipe heat exchanger.

II. Physical Description of Problem

The geometric configuration of grooved inner tube of double pipe heat exchanger investigated in this study is presented in Figure 1.

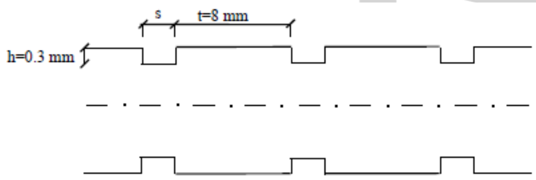

Fig. 1 Schematic Diagram of inner tube with groove of DPHE [2]

groove with rectangular surface and circumferential pattern are arranged on outer surface of inner tube side or in the annulus room of the double pipe heat exchanger. The groove depth (h) is taken 0.3, 0.5 and 1 mm; the distance between grooves (t) is 8 mm. The space of the groove (s) is 1 mm and 0.5 mm, respectively. The grooves are formed by conventional etching technique or by machining.

III. Mathematical Modeling and Methodology

The Navier-Stokes equations are the governing equations for fluid flow. They are a system of nonlinear partial differential equations. There are very few exact solutions of the equations, and even for simple geometries, the equations have to be solved numerically. With increasing the Reynolds number, the accurate solution of the equations becomes more difficult. A number of techniques and algorithms have been developed to obtain an accurate solution of the equation for high Reynolds numbers. The groove geometry is same as the lid driven cavity problem [9]. It is used typically to test new methods and codes.

We consider the incompressible viscous fluid flow and the Navier-Stokes equations in two space dimensions on rectangular domain = $[0, l_x] \times [0, l_y]$.

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{\partial p}{\partial x} + \mu\frac{\partial^2 u}{\partial x^2} + \mu\frac{\partial^2 u}{\partial y^2}$$

$$\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = -\frac{\partial p}{\partial y} + \mu\frac{\partial^2 v}{\partial x^2} + \mu\frac{\partial^2 v}{\partial y^2}$$

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

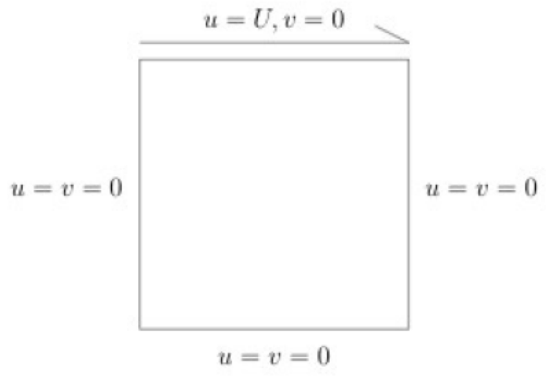

Fig. 2 Groove on inner tube

The four domain boundary conditions are denoted North, South, West and East. The domain is fixed in time, and we consider dirichlet boundary conditions. No-slip velocity boundary conditions (u=v=0) are applied on all walls except top lid. On top lid u=U and v=0 is applied.

Visulization

Every fixed number of time steps the current pressure and velocity field is visualized. The pressure field is shown as a color plot with some contour lines. On top of this, the normalized velocity field is shown as a quiver plot, i.e. little arrows indicate the direction of the flow. Since in a groove the flow rate varies significantly over different areas, out putting the normalized velocity field is a common procedure. Additionally, stream lines are shown. Those are paths particles would take in the flow if the velocity field was frozen at the current instant in time. Since the velocity field is divergence-free, the streamlines are closed. The stream lines are contour lines of the stream function q. It is a function whose orthogonal gradient is the velocity field.

$$(\nabla q)^{\perp} = u \quad \Leftrightarrow \quad -q_y = u \text{ and } q_x = v$$

Applying the 2d-curl to this equation yields

$$-\Delta q = \nabla \times (\nabla q)^{\perp} = \nabla \times u \quad \Leftrightarrow$$
$$-\Delta q = -\frac{\partial^2 q}{\partial y^2} - \frac{\partial^2 q}{\partial x^2} = \frac{\partial u}{\partial y} - \frac{\partial v}{\partial x}$$

The stream function exists since the compability condition $\nabla . u = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$ is satisfied.

Discretization schemes and algorithm

Convection terms are discretized with UPWIND scheme. Diffusion terms are discretized with central differencing scheme. SIMPLER Algorithm is used for solving the equations. A mesh of 90X90 size is used to solve the

 DOI : 10.18231/2454-9150.2018.0198

standard case with Reynolds number 100, 1000, 10000 and groove depth 0.3, 0.5 and 1 mm using MATLAB. The MATLAB code provides variable box sizes, grid resolution, time step, and Reynolds numbers. It provides implicit time stepping for the fluid viscosity. The visualization of pressure field, velocity field and streamlines is possible. It also provides automatic transition from central to donor-cell discretization for the nonlinear advection part.

In order to ensure the accuracy of the numerical results, the mesh sensitivity test was conducted for groove width and depth = 1. A series of grid sensitivity tests were carried out to ensure that optimized computational mesh was obtained.

Annulus side velocity, Reynolds number and Nusselt Number

The mean annulus side fluid velocity is defined by:

$$u_a = \frac{m_a}{\rho_a A_{cross}}$$

Where m_a is the mass flow rate, ρ_a is air density in the annulus side and A_{cross} represents the characteristic cross-section area. For a helically baffles annulus side:

$$A_{cross} = 0.5\,(D_{io} - D_{oi})$$

Where, D_{io} is the internal diameter of the outer pipe, D_{oi} is the external diameter of the inner tube. Here, we ignored the change in characteristic cross area due to groove.

With the mean velocity value, the Reynolds number for the annulus side is determined by:

$$Re = \frac{\rho_a u_a D_h}{\mu_a}$$

With μ is the dynamic viscosity of fluid in the annulus side. $D_h = (D_{io}-D_{oi})$ is equivalent hydraulic diameter.

Heat transfer rate of the annulus side fluid:

$$Q_a = m_a c_{pa}\left(T_a^{out} - T_a^{in}\right)$$

Heat transfer rate of the tube side fluid:

$$Q_t = m_t c_{pt}\left(T_t^{out} - T_t^{in}\right)$$

Where, m_a is the flow rate, T the temperature, c_p the specific heat. The subscript a and t refers respectively to the annulus side and inner tube side; superscript 'in' and 'out' stand for the values at the inlet and outlet respectively. The physical properties are evaluated at the average temperature of the inlet and outlet for each side of the heat exchanger.

Heat transfer coefficient of annulus side is:

$$h = Q_a / A(T_s - T_{fm})$$

Where, Ts is inner tube (hot) surface temperature. T_{fm} is the bulk mean air temperature, calculated as

$$T_{fm} = (T_o + T_i)/2$$

The Nusselt number for the annulus side is:

$$Nu = \frac{h_a D_h}{k}$$

Where, h_a is the annulus side heat transfer coefficient, D_h equivalent hydraulic diameter, k the thermal conductivity of the fluid.

IV. RESULTS AND DISCUSSION

A study is carried in groove on inner tube with varying aspect ratio and Reynolds number. It is found that streamlines are highly dependent on aspect ratio and Reynolds number. The standard case is taken as Reynolds number 1000 with an aspect ratio of 0.5. Here we have one primary vortex and two corner vortices (Fig 7). As the Reynolds number increases the vortices are being steep and displaced towards its right side upper corner and hence fluid inside the groove try to displace in main flow causes its diffusion. Diffusion of such vortices i.e. eddies in main flow causes strong mixing of fluid which increases the local temperature which leads to increase of final outlet temperature. Form the fundamental equations it is clearly shows that there will be rise in bulk mean temperature leads to decrease of difference between Ts, inner tube (hot) surface temperature and $T_{fm,}$ the bulk mean air temperature. So finally we can say that heat transfer coefficient will increase due to turbulence. The main observation of the study with aspect ratio 2 is that at bottom there is vacant space in the groove as far as contours is concern (Fig. 12). As the Reynolds number increases the main contour displaced towards right side top corner of groove irrespective of aspect ratio (Fig. 3-11) The Boundary layer become thinner at right side of groove with increase of Reynolds number.

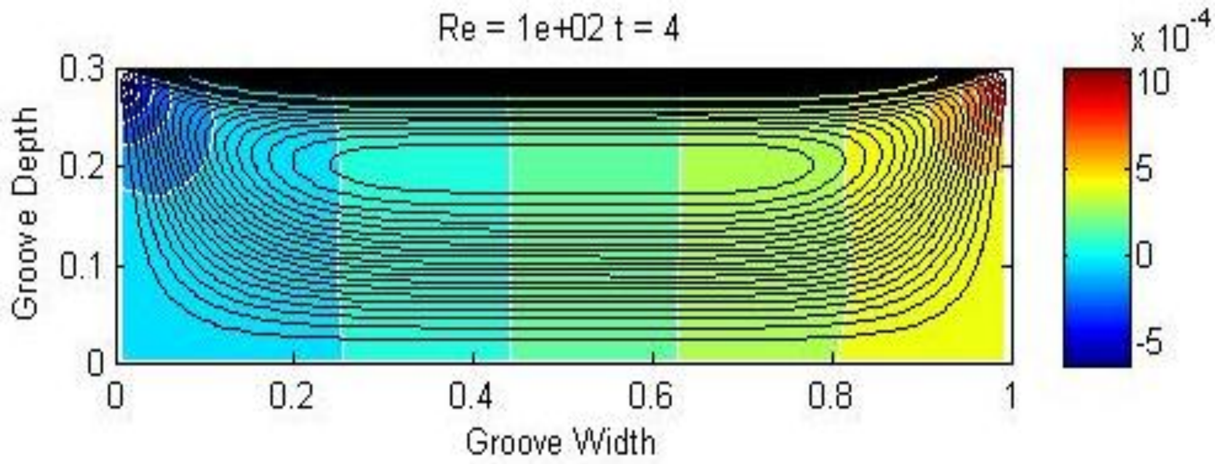

Fig. 3 Contours for Re=100 and Aspect Ratio = 0.3

 DOI : 10.18231/2454-9150.2018.0198

ISSN : 2454-9150 Vol-04, Issue-02, May 2018

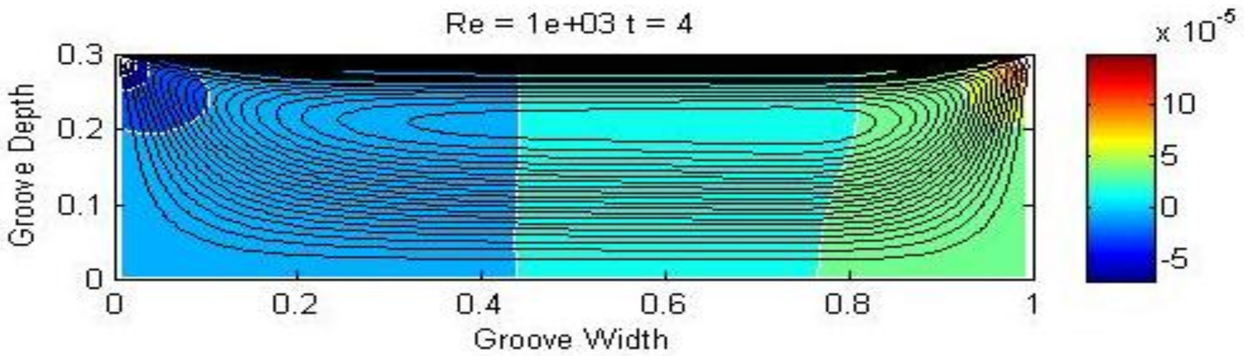

Fig. 4 Contours for Re=1000 and Aspect Ratio = 0.3

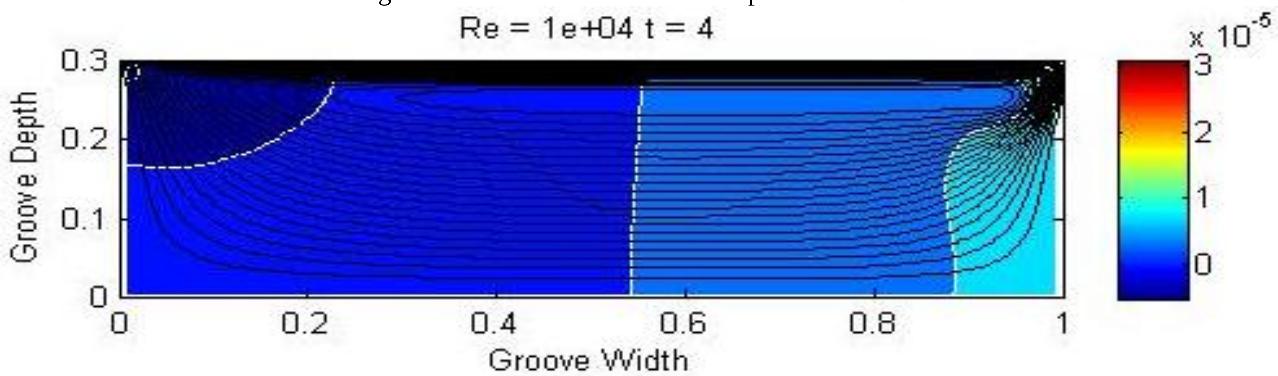

Fig. 5 Contours for Re=10000 and Aspect Ratio = 0.3

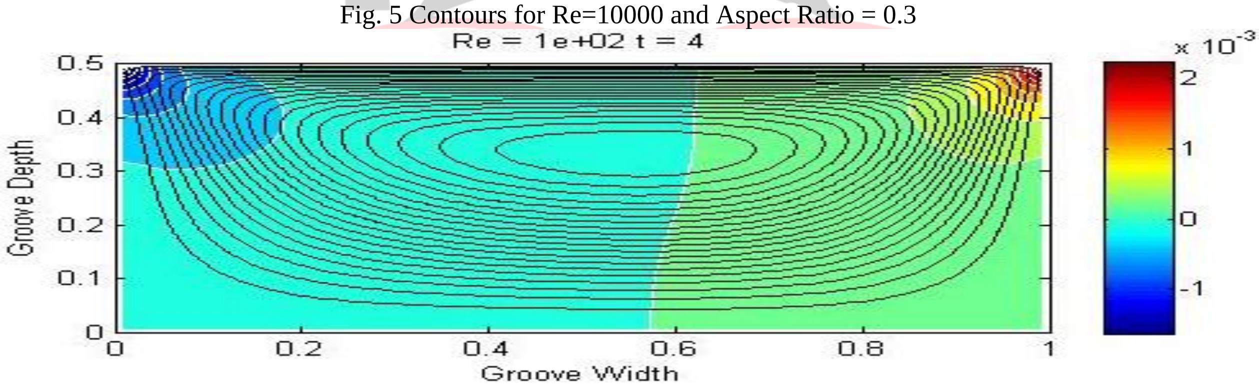

Fig. 6 Contours for Re=100 and Aspect Ratio = 0.5

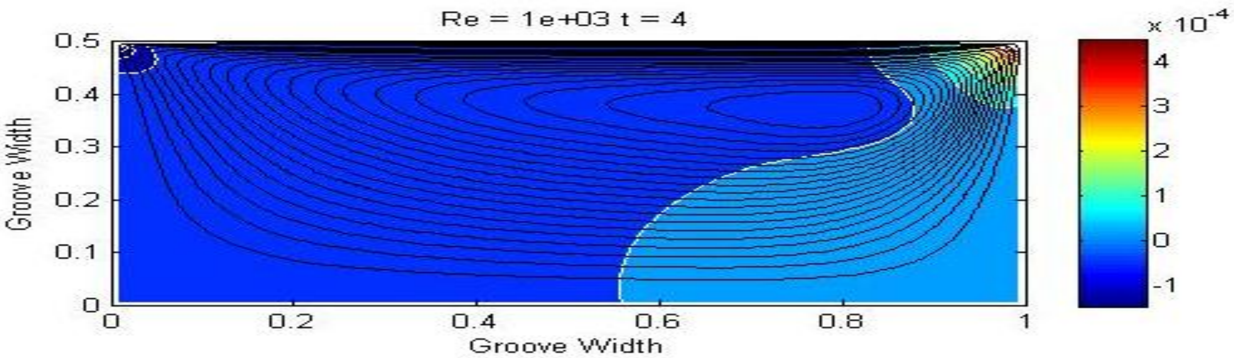

Fig. 7 Contours for Re=1000 and Aspect Ratio = 0.5

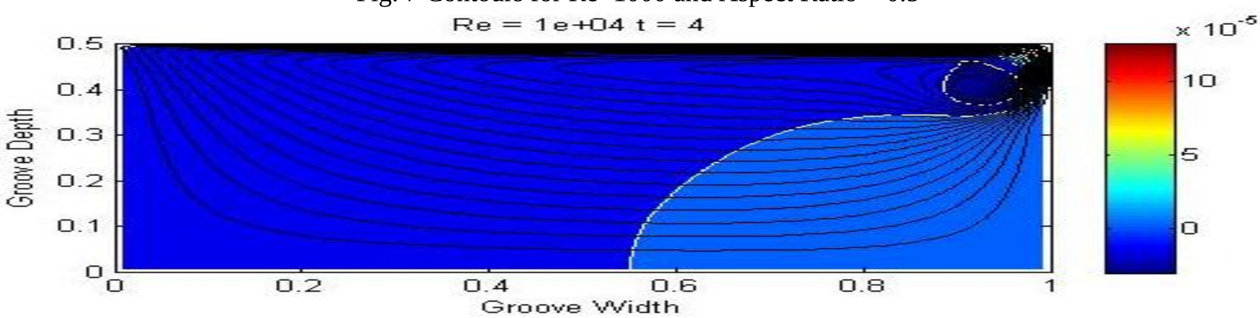

Fig. 8 Contours for Re=10000 and Aspect Ratio = 0.5

 DOI : 10.18231/2454-9150.2018.0198

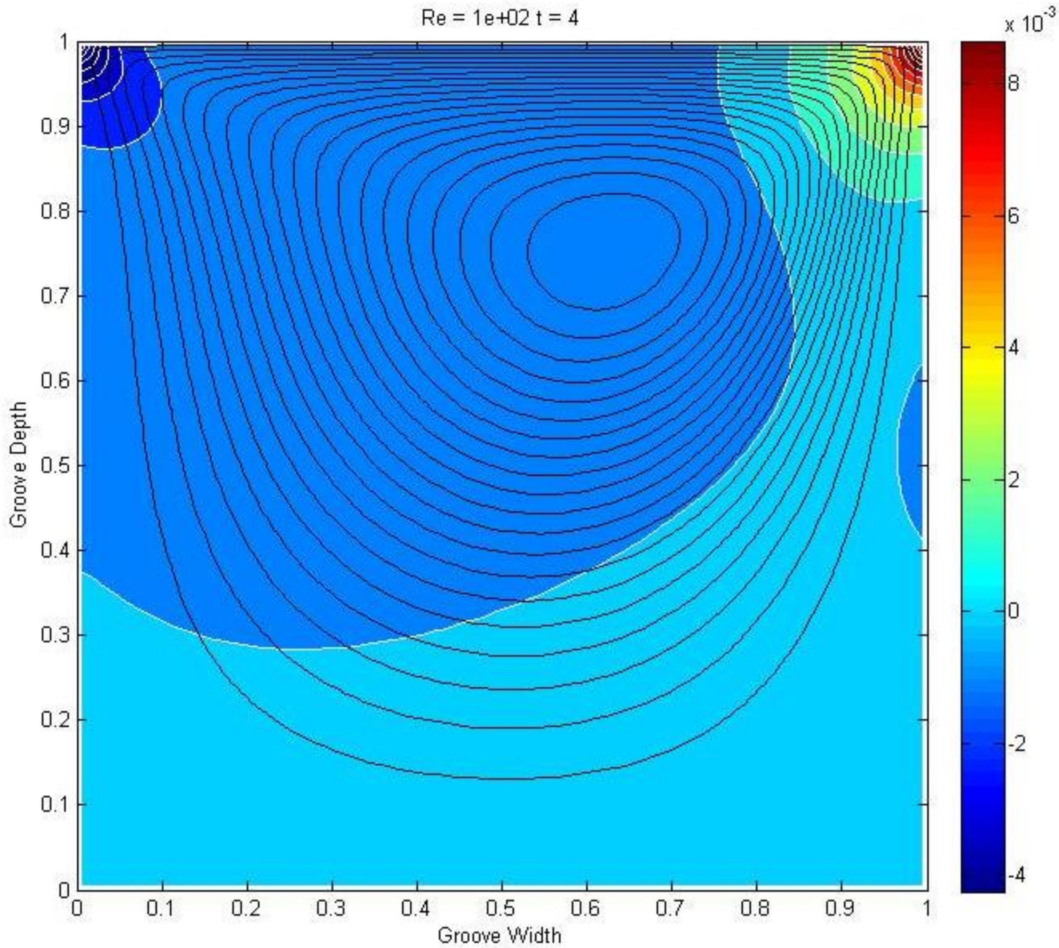

Fig. 9 Contours for Re=100 and Aspect Ratio = 1

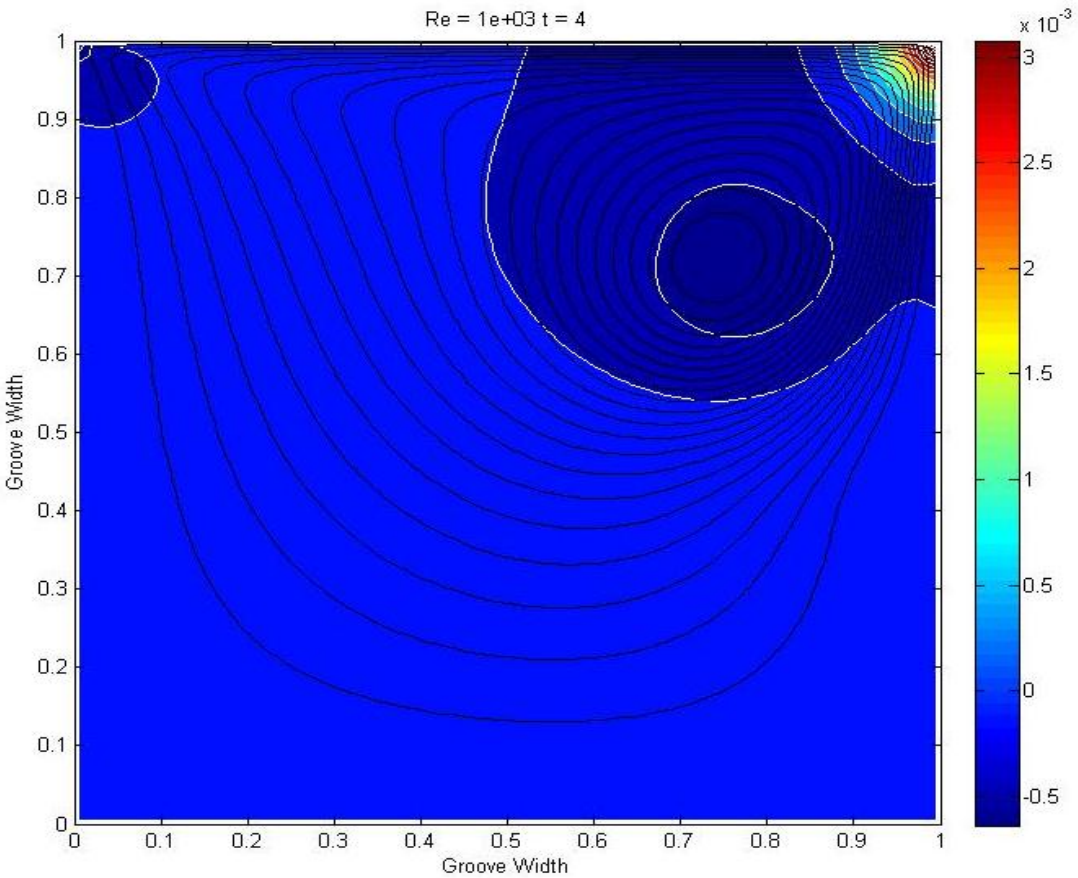

Fig. 10 Contours for Re=1000 and Aspect Ratio = 1

 DOI : 10.18231/2454-9150.2018.0198

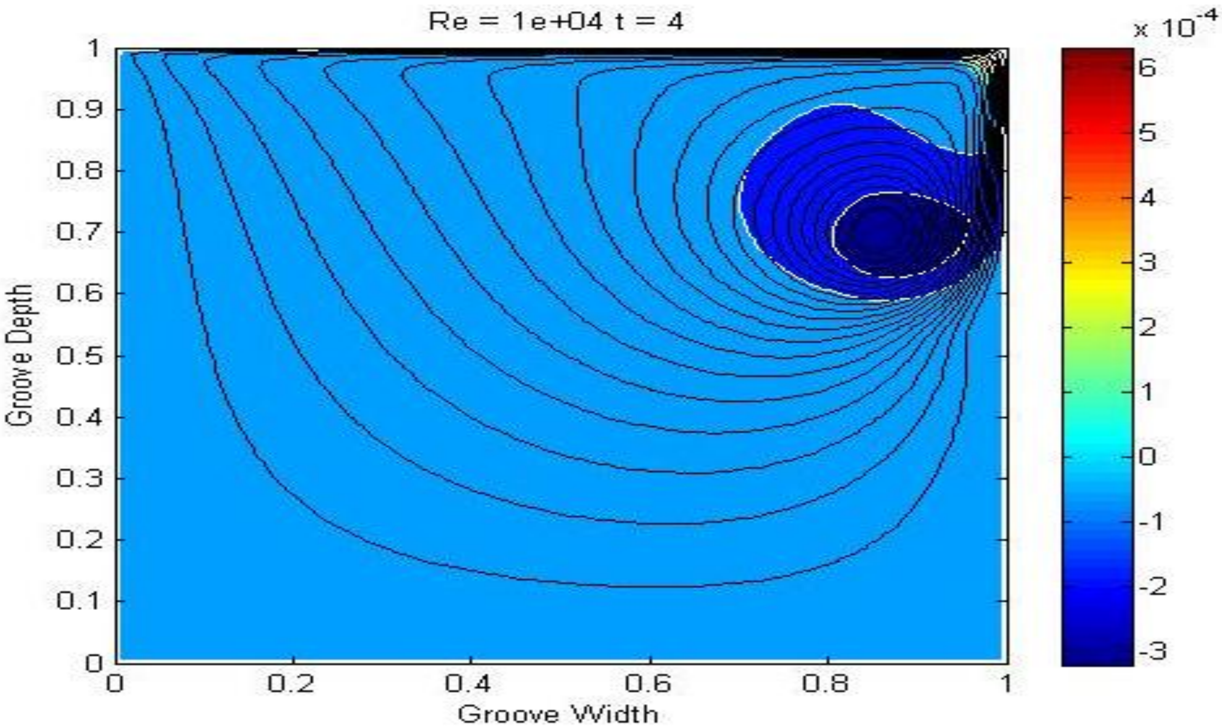

Fig. 11 Contours for Re=10000 and Aspect Ratio = 1

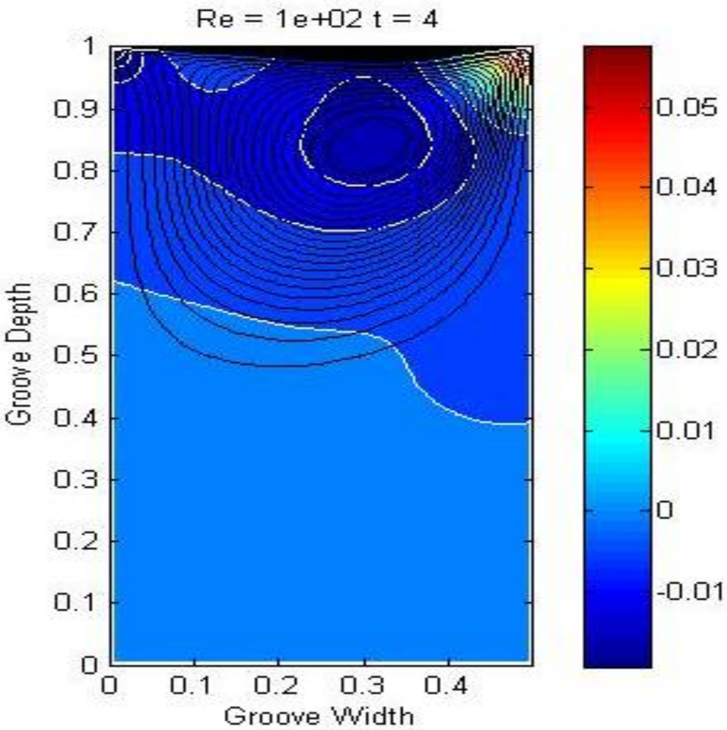

Fig. 12 Contours for Re=100 and Aspect Ratio =2

V. CONCLUSION

Numerical simulations are carried out for laminar incompressible fluid flow in groove for different aspect ratio and different Reynolds number. The nature of the flow inside groove cavity of different aspect ratio is visualized. It is found that the dynamics and structure of primary vortex are strongly affected by the Reynolds number as well as the aspect ratio of the groove.

The groove on the inner pipe is responsible to produce near wall turbulence with vortices inside the groove which increase the pressure drop to fluid flow. The turbulence is also responsible to increase the local fluid temperature and finally it will increase the bulk mean temperature of fluid flowing in annular passage of double pipe heat exchanger. Hence, we may say that the groove on tube for improvement of heat transfer rate are effective but with certain pressure loss penalty. The performance prediction of double pipe heat exchanger (DPHE) with groove is the outcome of this paper.

REFERENCES

[1] Sunu, Putu Wijaya, Simon Anakottapary Daud, and Wayan G. Santika. "Temperature approach optimization in the double pipe heat exchanger with

 DOI : 10.18231/2454-9150.2018.0198

groove." MATEC Web of Conferences. Vol. 58. EDP Sciences, 2016.

[2] Sunu, Wijaya, and Made Rasta. "Heat Transfer Enhancement and Pressure Drop of Grooved Annulus of Double Pipe Heat Exchanger." (2017).

[3] Sunu, Putu Wijaya et. al. . "Flow behavior and friction factor in internally grooved pipe wall." Adv. Studies Theor. Phys 8.14 (2014): 643-647.

[4] Sunu, Putu Wijaya, et al. "The effect of wall groove numbers on pressure drop in pipe flows." International Journal of Fluid Mechanics Research 42.2 (2015).

[5] Sunu, Putu Wijaya. "The characteristics of increased pressure drop in pipes with grooved." Adv. Studies Theor. Phys 9.2 (2015): 57-61.

[6] Sunu, Putu Wijaya, et al. "Optimal grooves number for reducing pressure drop." Contemporary Engineering Sciences9.22 (2016): 1067-1074.

[7] Sunu, P. W., et al. "Experimental Studies on Grooved Double Pipe Heat Exchanger with Different Groove Space." Journal of Physics: Conference Series. Vol. 953. No. 1. IOP Publishing, 2018.

[8] Kuber Dwivedi et. al. “The Heat Transfer Enhancement Techniques-A Review.” Nt. Conf. on Comp. and Exp. Methods in Mechanical Engineering: GLBITM, Noida, 2017.

[9] Al-Amiri, Abdalla M., Khalil M. Khanafer, and Ioan Pop. "Numerical simulation of combined thermal and mass transport in a square lid-driven cavity." International journal of thermal sciences 46.7 (2007): 662-671.

 DOI : 10.18231/2454-9150.2018.0198