RESEARCHERID
THOMSON REUTERS
[Ashok et al., 6(7), July 2021]

ISSN : 2455-9679
Impact Factor : 3.805

ijrtsm

IJRTSM

INTERNATIONAL JOURNAL OF RECENT TECHNOLOGY SCIENCE & MANAGEMENT

"CFD ANALYSIS ON SHELL AND TUBE TYPE HEAT EXCHANGER BY USING DIFFERENT BAFFLE INCLINATION ANGLE"

Ashok Kumar [1], *Neeraj Yadav* [2]

[1] *P. G. Scholar, Department of Mechanical Engineering, BERI Bhopal, Madhya Pradesh, India*

[2] *Assistant Professor, Department of Mechanical Engineering, BERI Bhopal, Madhya Pradesh, India*

ABSTRACT

Heat exchangers are one of the for the most element applied hardware inside the system ventures. Warmth exchangers are utilized to transport warmness between two interplay streams. Here modeling of heat exchanger has done on CATIA software and simulation has performed on ANSYS CFD platform . Hence the design can be changed for higher warmness switch in approaches both the decreasing the shell diameter, which will be a right touch with the baffle or by using growing the baffle in order that baffles could be right touch with the shell. It's miles due to the fact the heat switch region isn't utilized correctly. So we are able to visible that when angle of inclination baffle might be extended then heat transerferd we located that most. Right here we use 180 ^{0} and 10 ^{0} degree inclination attitude with baffles and 10 degree inclination baffles we determined most heat transfer examine to 180 ^{0} degree inclination of baffles.

Key Words: *CFD, ANSYS , CATIA, heat exchanger, heat, inclination.*

I. INTRODUCTION

Heat exchanger is a device that continuously transfers heat from one medium to other medium in order to carry process energy. The purpose of constructing a heat exchanger is to get an efficient method of heat transfer from one fluid to another, by direct contact or by indirect contact. Heat exchangers are one of the mostly used equipment in the process industries. Heat exchangers are used to transfer heat between two process streams. One can realize their usage that any process which involve cooling, heating, condensation, boiling or evaporation will require a heat exchanger for these purpose. Process fluids, usually are heated or cooled before the process or undergo a phase change. Different heat exchangers are named according to their application. For example, heat exchangers being used to condense are known as condensers, similarly heat exchanger for boiling purposes are called boilers. Shell and tube type heat exchanger consists of a number of tubes through which one fluid flows. Another fluid flows through the shell which encloses the tubes and other supporting items like baffles, tube header sheets, gaskets etc. The heat exchange between the two fluids takes through the wall of the tubes.

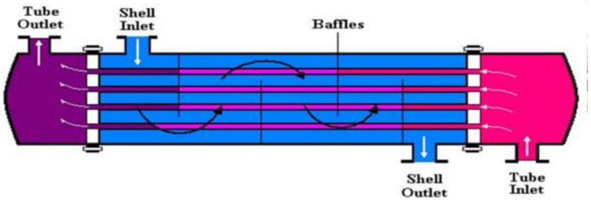

Fig.1.1 Heat Exchanger

ISSN : 2455-9679
Impact Factor : 3.805

II. METHODOLOGY

CFD is a sophisticated computationally-based design and analysis technique. CFD software gives you the power to simulate flows of gases and liquids, heat and mass transfer, moving bodies, multiphase physics, chemical reaction, fluid-structure interaction and acoustics through computer modeling. This software can also build a virtual prototype of the system or device before can be apply to real-world physics and chemistry to the model, and the software will provide with images and data, which predict the performance of that design.

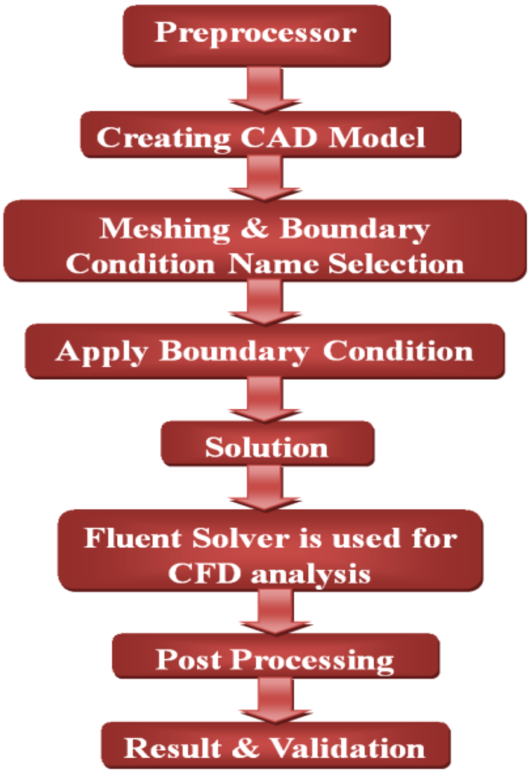

Fig.2.1 Methodology

CFD analysis of helical coil heat exchanger using Ansys 14.0, Cad model Generation of 3D model by using CATIA Ver 5.0 and exporting to the IGES. and then import in ANSYS fluent 14.0.

➢ **PRE PROCESSING**:

Create geometry and mesh for solving problem

❖ **CAD model**

✓ Generation of 3D model by using CATIA ver 5.0

❖ **FVM approach:-**

By this method we can solve algebraic equation to get initial solution.

✓ Set the transportation equation that needs to be solved. This can be solved by using ANSYS Fluent 14.0 in Fluent setup
✓ Set the fluid property
✓ Set the boundary conditions
✓ Set the Source term (Pressure)

➢ **SOLUTION:**

Solution Method

❖ **Pressure Velocity coupling scheme**

For 2D Problem we use Stream Function Vortices approach.

For 3D Problem we use Primitive variable approach.

ISSN : 2455-9679
Impact Factor : 3.805

Collocated grid should be used to solve pressure velocity coupling scheme.

- **Turbulence Modeling:-**
  K-ε turbulence model for turbulent flow equation Momentum second order Turbulence Kinetic energy second order (K)

- Turbulence dissipation rate second order(ε)

- **SOLUTION INITIALIZATION:-**

Initialized the solution to get the initial solution for the problem.

By using SIMPLE solver (Semi – implicit method for pressure linked equation).

- **RUN SOLUTION:-**

Run the solution by giving 300 no of iteration for solution to converge.

- **POST PROCESSING:-**

- Post Processing:- For viewing and interpretation of Result, the result can be viewed in various formats like graph, value, animation etc.

Table 2.1 Geometric dimensions of shell and tube heat exchanger

Heat exchanger length, *L*	600mm
Shell inner diameter, *Di*	90mm
Tube outer diameter, do	20mm
Tube bundle geometry and pitch	30mm
Number of tubes, *Nt*	5
Number of baffles. *Nb*	5
Central baffle spacing, *B*	86mm
Baffle inclination angle, θ	180^0 to 10^0

Table 2.2 Boundary Condition shell and tube heat exchanger

Inlet Temperature of Shell Side (K)	353 (K)
Inlet Temperature of Tube Side (K)	300 K
Turbulence intensity ratio	0.5 %
Turbulence viscosity ratio	10 %
Constant wall temperature (T_c)	293 K
Outlet ($P_{out.}$)	Pressure outlet
Operating condition pressure ($P_{opr.}$)	101325 Pa

ISSN : 2455-9679
Impact Factor : 3.805

III. SIMULATION

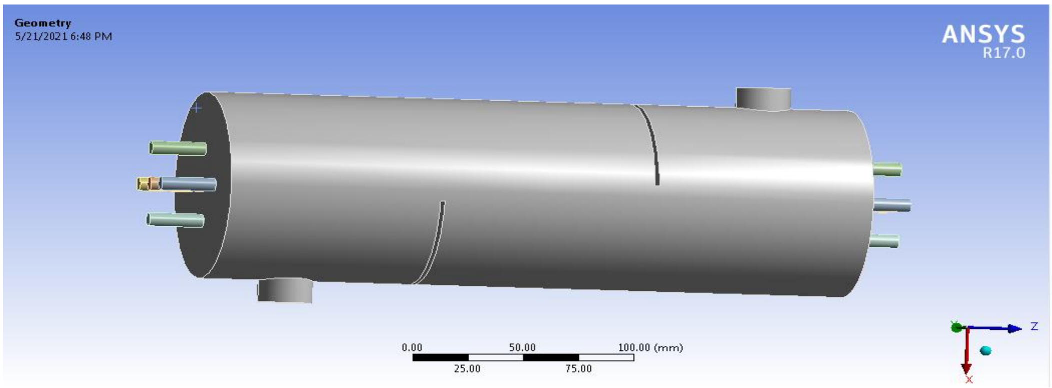

Fig.3.1 Complete model of shell and tube heat exchanger

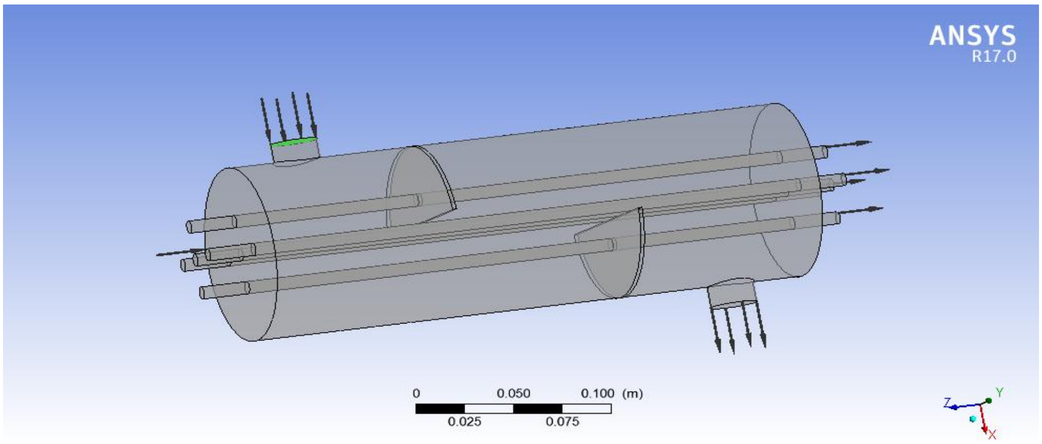

Fig 3.2 Isometric view of arrangement of baffles and tubes of shell and tube heat exchanger with baffle inclination

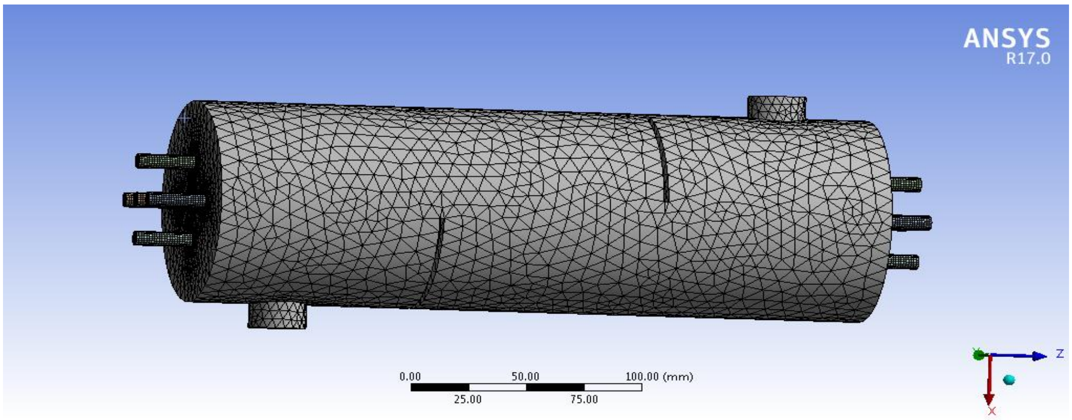

Fig 3.3 Complete model of shell and tube heat exchanger meshing

ISSN : 2455-9679
Impact Factor : 3.805

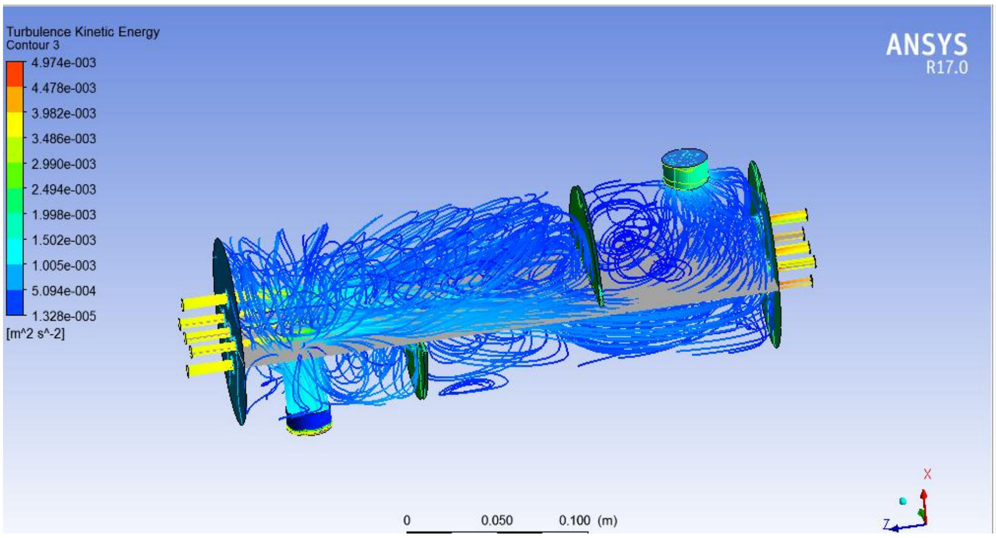

Fig 3.4 Heat exchanger model in CFX fluid flow Turbulence kinetic energy 180° inclinations

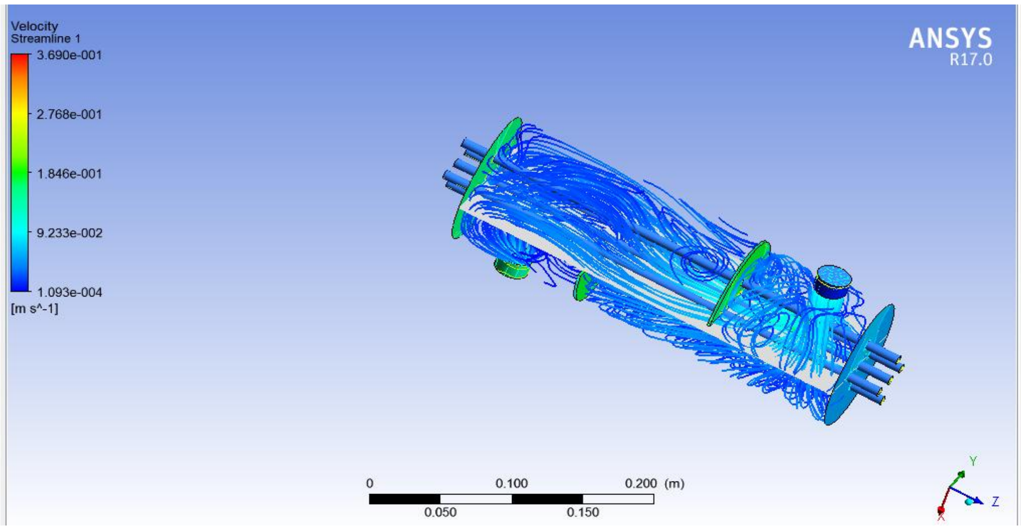

Fig 3.5 Heat exchanger model in CFX fluid flow velocity 180° inclinations

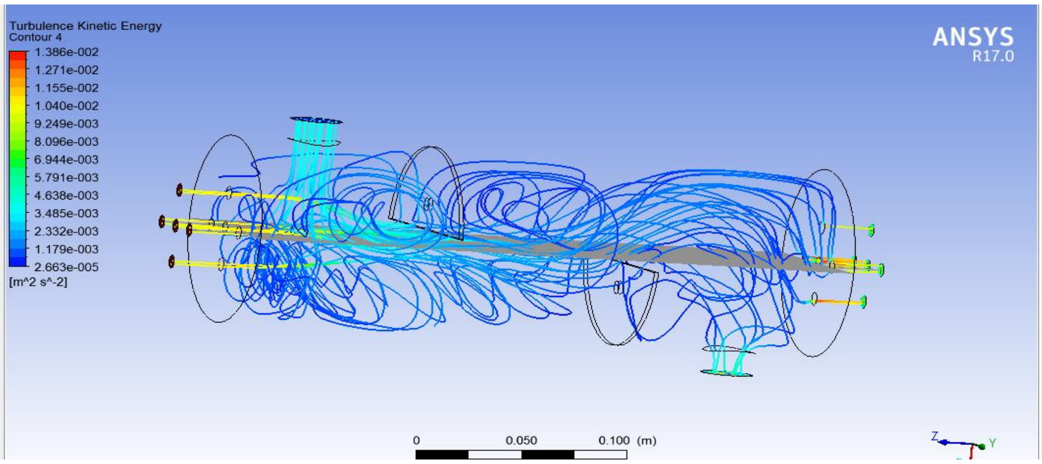

Fig.3.6 Heat exchanger model in CFX fluid flow Turbulence Kinetic Energy 10° inclinations

ISSN : 2455-9679
Impact Factor : 3.805

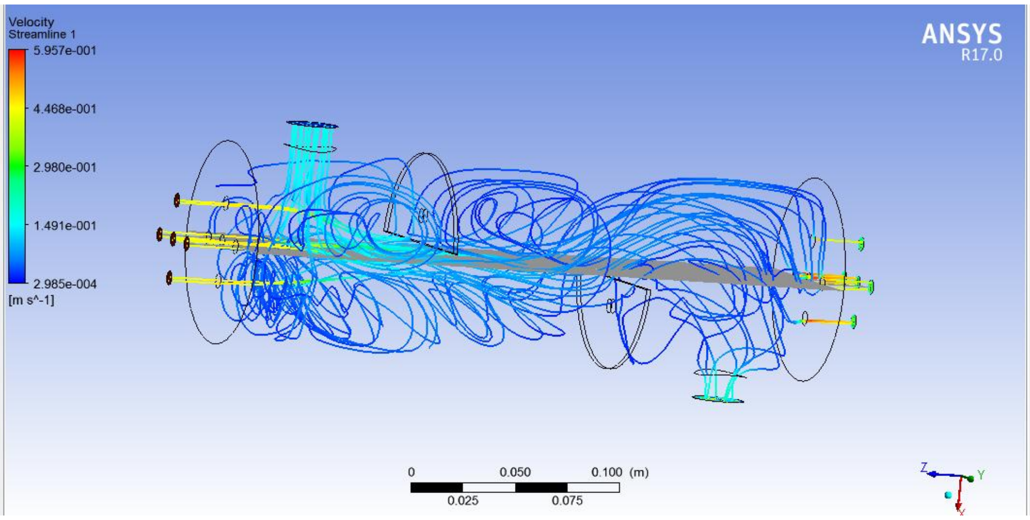

Fig.3.7 Heat exchanger model in CFX fluid flow velocity 10° inclinations

IV. RESULT

Table 4.1 Plot of Baffle inclination angle vs Outlet Temperature of shell and tube side

Baffle Inclination (In Degree)	Shell Inlet Temperature (K)	Shell Outlet Temperature (K)	Tubes Inlet Temperature (K)	Tubes Outlet Temperature (K)
180^0	353	348.6	300	321.5
10^0		347.6	300	314.9

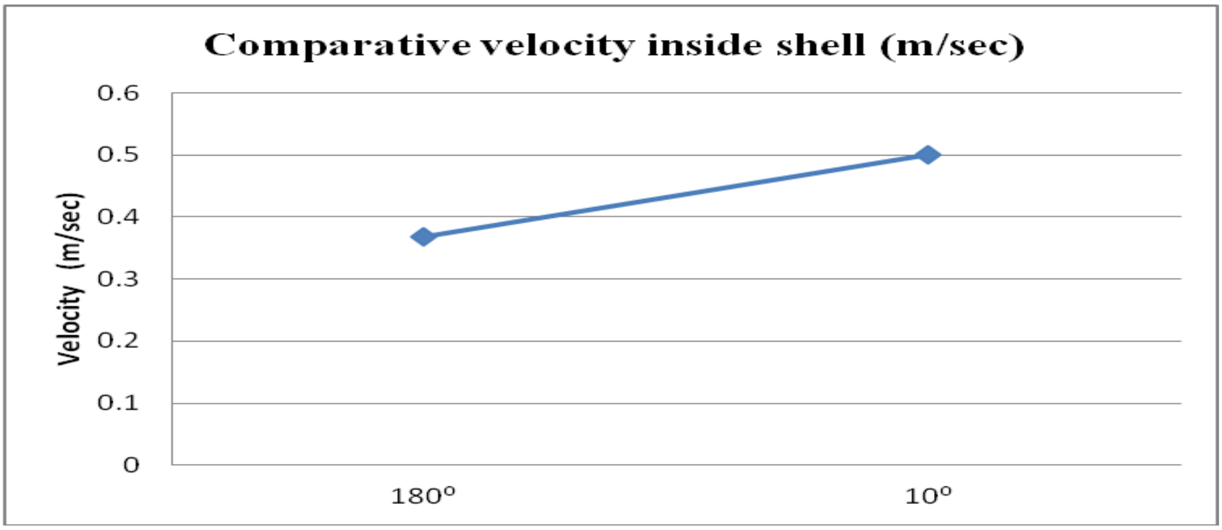

Fig.4.1 Plot of Velocity profile inside shell

ISSN : 2455-9679
Impact Factor : 3.805

V. CONCLUSION

The Heat Exchanger and go with the flow distribution is discussed in element and proposed version is in comparison this version also can be progressed by using the usage of nusselt variety and Reynolds pressure model, but with better computational concept. Furthermore the decorate wall feature are not use on this undertaking, however they may be very useful. The warmth switch is poor due to the fact most of the fluid passes with out the interplay with baffles. Hence the design can be changed for higher warmness switch in approaches both the decreasing the shell diameter, which will be a right touch with the baffle or by using growing the baffle in order that baffles could be right touch with the shell. It's miles due to the fact the heat switch region isn't utilized correctly. So we are able to visible that when angle of inclination baffle might be extended then heat transerferd we located that most. Right here we use 180^0 and 10^0 degree inclination attitude with baffles and 10 degree inclination baffles we determined most heat transfer examine to 180^0 degree inclination of baffles.

REFERENCES

1. A.O. Adelaja, S. J. Ojolo and M. G. Sobamowo, "Computer Aided Analysis of Thermal and Mechanical Design of Shell and Tube Heat Exchangers", Advanced Materials Vol. 367 (2012) pp 731-737 © (2012) Trans Tech Publications, Switzerland.

2. Yusuf Ali Kara, Ozbilen Guraras, "A computer program for designing of Shell and tube heat exchanger", Applied Thermal Engineering 24(2004) 1797–1805.

3. Rajagapal THUNDIL KARUPPA RAJ and Srikanth GANNE, "Shell side numerical analysis of a shell and tube heat exchanger considering the effects of baffle inclination angle on fluid flow", Thundil Karuppa Raj, R., et al.: Shell Side Numerical Analysis of a Shell and Tube Heat Exchanger ,THERMAL SCIENCE: Year 2012, Vol. 16, No. 4, pp. 1165-1174.

4. S. Noie Baghban, M. Moghiman and E. Salehi, " Thermal analysis of shell-side flow of shell-and tube heat exchanger using experimental and theoretical methods" (Received: October 1, 1998 - Accepted in Revised Form: June 3, 1999).

5. A.GopiChand, Prof.A.V.N.L.Sharma , G.Vijay Kumar, A.Srividya, " Thermal analysis of shell and tube heat exchanger using mat lab and floefd software", Volume: 1 Issue: 3 276 – 281, ISSN: 2319 – 1163.

6. Hari Haran, Ravindra Reddy and Sreehari, "Thermal Analysis of Shell and Tube Heat Exchanger Using C and Ansys" , International Journal of Computer Trends and Technology (IJCTT) – volume 4 Issue 7–July 2013.

7. J. Li and G.P. Peterson, "Experimental measurements of fluid flow and heat transferring micro-channel cooling passages in a chip substrate, Proceedings of the ASME International Electronics Packaging Conference in Binghamton, NY, USA, ASME publications, 1993, Vol. 4-2, p. 685–692.

8. Pfund, D., Rector, D., Shekarriz, A., Popescu, A., and Welty, J., "Pressure drop measurements in a micro channel", AICHE Journal, 2000, Vol. 46(8), p. 1496–1507.

9. Rahman, Vafai ,M.M. and Gui, F., Design, fabrication, and testing of micro channel heat sinks for aircraft avionics cooling", Proceedings of the Intersociety Energy Conversion Engineering Conference, 1993, Vol. 1, p. 1–6.

10. Wei and Joshi, "Enhanced heat transfer in the entrance region of micro channels ",Proceedings of the Intersociety Energy Conversion Engineering Conference, 1995, p. 289–294.

11. Gui and Anderson, "Single-phase liquid friction factors in micro channels ",International Journal of Thermal Sciences, 2005

ISSN : 2455-9679
Impact Factor : 3.805

12. Davies, "An experimental study of convective heat transfer in silicon micro channels with different surface conditions", International Journal of Heat and Mass Transfer, 2003.

13. Pucha, X.F., Wang, B.X., Peterson, G.P., and Ma, H.B., "Experimental investigation of heat transfer in flat plates with rectangular micro channels", International Journal of Heat and Mass Transfer, 1995, Vol. 38(1), p. 127–137.

14. Zhao, X.F. and Peterson, G.P., "Effect of thermo fluid and geometrical parameters on convection of liquids through rectangular micro channels", International Journal of Heat and Mass Transfer, 1995, Vol. 38(4), p. 755–758

15. Donald Q.Kern. 1965. Process Heat transfer (23rd printing 1986). McGraw-Hill companies. ISBN 0-07-Y85353-3.

16. Richard C. Byrne Secretary. 1968. Tubular Exchanger Manufacturers Association, INC. (8th Edition). 25 North Broadway Tarrytown, New York 10591.

17. R.H Perry. 1984. Perry's Chemical Engineer's Handbook (6th Edition ed.). McGraw-Hill. ISBN 0-07-049479-7.