International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 08 Issue: 01 | January - 2024 SJIF Rating: 8.176 ISSN: 2582-3930

Computational Fluid Dynamics (CFD) Analysis of the Shell and Tube Heat Exchanger by Use of Different NanoFluids

P.Surya[1], Dr. Siva Naga Malleswara Rao Singu[2], M.VishnuPrasad[3], S. Durga Prasad[4]

[1]Assistant Professor, Department of Mechanical Engineering & Srinivasa Institute of Engineering and Technology, Amalaparum, India.
[2]Project Manager, Vensai Technologies, GA, USA[2]
[3,4]Assistant Professor, Department of Mechanical Engineering & Srinivasa Institute of Engineering and Technology, Amalaparum, India.

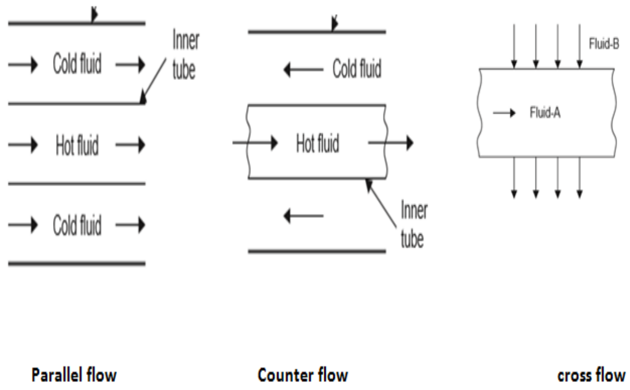
---***---

Abstract - Heat exchanger is a device used to transfer heat between one or more fluids. The fluids may be separated by a solid wall to prevent mixing or they may be in direct contact. These exchangers provide true counter-current flow and are especially suitable for extreme temperature crossing, high pressure, high temperature, and low to moderate surface area requirements different Nano partials mixed with base fluids is called Nano fluids and analysed for their performance of Nano fluids by use in the heat exchanger. The Nano fluids are Aluminium Oxide, Silicon Oxide and Titanium carbide.The volume concentration of the nanoparticle use in this study is 0.03% and mass flow rate 8 lpm,Nano fluid inlet temperature 333k and normal water fluid inlet temperature 300k. 3D model of the compact shell and tube heat exchanger is done in CATIA V5 and CFD analysis is done on the shell and tube heat exchanger by using ANSYS 15.0 fluent work bench. Compare three Nano fluids values for better Nano fluid choose one.

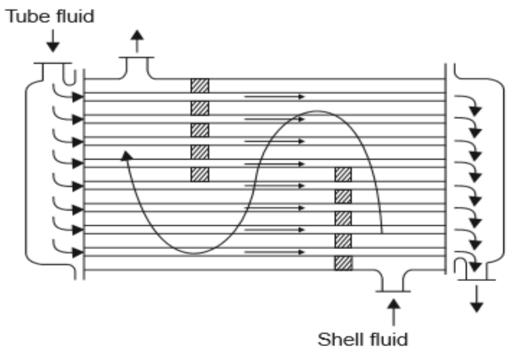
Key Words**:** Catia, cfd, Nano fluids.

1.INTRODUCTION

Heat exchanger is a device used to transfer heat between one or more fluids. The fluids may be separated by a solid wall to prevent mixing or they may be in direct contact. These exchangers provide true counter-current flow and are especially suitable for extreme temperature crossing, high pressure, high temperature, and low to moderate surface area requirements different Nano partials mixed with base fluids is called Nano fluids and analysed for their performance of Nano fluids by use in the heat exchanger. The Nano fluids are Aluminium Oxide, Silicon Oxide and Titanium carbide.The volume concentration of the nanoparticle use in this study is 0.03% and mass flow rate 8 lpm,Nano fluid inlet temperature 333k and normal water fluid inlet temperature 300k. 3D model of the compact shell and tube heat exchanger is done in CATIA V5 and CFD analysis is done on the shell and tube heat exchanger by using ANSYS 15.0 fluent work bench. Compare three Nano fluids values for better Nano fluid choose one.



Shell and tube arrangement: Single tube arrangement is suitable only for very small capacities. So the single tube type is not extensively used in industry. Shell and tube type is the most popular arrangement. A number of small bore pipes are fitted between two tube plates and one fluid flows through these tubes. The tube bundle is placed inside a shell and the other fluid flows through the shell and over the surface of the tubes



NANOFLUID

A Nano fluid is a fluid containing nanometre sized particles called nanoparticalsThe Nano particles used in Nano fluid are typically made of metals, oxides, carbides (or) carbon nanotubes

PROPERTY'S	ALUMINUM OXIDE	SILICON OXIDE	TITANIUM CARBIDE
Thermal conductivity(K) W/M-K	30	5.4	11.8
Density Kg/m3	3970	2400	4260
specific heat(Cp) J/kg.k	765	670	683

II. DESIGN PROCEDURE

CATIA

CATIA is an acronym for Computer Aided Three-dimensional Interactive Application. It is one of the leading 3D software used by organizations in multiple industries ranging from aerospace, automobile to consumer products. It helps to create 2D or 3D solid models without any complexity and easily to edit the design at any stage in the design process. Model created by catia software.

ANSYS (CFD Analysis by use fluent tool)

STEPS IN ANSYS

Define Material Properties : Now that the part exists, define a library of the necessary materials that compose the object (or project) being modeled. This includes thermal and mechanical properties.

Generate Mesh : At this point ANSYS understands the makeup of the part. Now define how the modeled system should be broken down into finite pieces.

Apply boundary condition: Once the system is fully designed, the last task is to burden the system with constraints, such as physical loadings or boundary conditions.

Obtain Solution :This is actually a step, because ANSYS needs to understand within what state (steady state, transient… etc.) the problem must be solved.

Present the Results: After the solution has been obtained, there are many ways to present ANSYS' results, choose from many options such as tables, graphs, and contour plots.

III. LITERATURE SURVEY

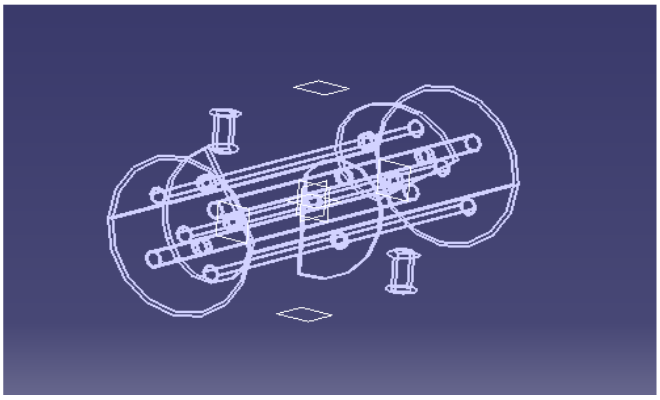
VishnuPrasad Mahanthi (2O21) It'svery easy to build a double pipe heat exchanger. In another it uses a heat exchanger pipe.After the required thermal exchanger has been decided, the pipe dimensions and turning numbers can bechosen surface area for counter flow or parallel flow.The type of flow pattern is an important element in the design of the dual heat exchanger. Usually, a doubleheat exchanger is a counterblow or parallel flow. Cross flow is just not working with a heat exchanger dual tubing. The flow pattern and heat exchange duty needed allow the measurement of the difference in the mean log temperature. In conjunction with an approximate overall coefficient of heat transfer, the appropriate surface area can be calculated. It is then possible to calculate the tube sizes, tube length and number of bends.

The concentrations of Fe3O4 Nano fluids in a double-pipeline interchange rope tubes with a return curve were evaluated experimentally and turbulent flow conditions for Convective heat transmission, friction factor and efficiency of various volume levels.The hot nanoplanets circulate though an internal pipe, while a nanoplane pipe passes the cold water. The nanoparticles in this sample have concentrations of 0.03 and 8 lpm, with mass flow rates of 10 lpm and numbers of Reynolds ranging from 9,000 to 30,000. CFD analysis is done in the Ansys Fluent 15.0 workbench and the various nanofluids used in the procedure analyse, which are based on the effects of nanofluids with a higher heat transfer rate. Different nanofluids (Al2o3, Cuo, Fe3o4) have been used in this operation, and different basic paper properties are used.

VishnuPrasad Mahanthi (2O24) In thermal and nuclear power plants, the chemical and process industries, house and ocean depths, and fluid supply techniques in industries, pressure vessel cylinders find extensive applications. The permissible pressure for the weld force, represented as weld efficiency, is just right design follow. The ratio of the longitudinal (axial) force of a welded joint to the longitudinal force of the pipe or tank shell is known as efficiency. This thesis uses the ANSYS finite element analysis programme to analyse the pressure vessel's strength and design it based on the weld efficiency. When designing a pressure vessel, mathematical correlations will likely be taken into consideration. The organisation will target the design parameters based on the desired weld efficiency. CATIA will be used to complete the modelling. An ANSYS analysis of the pressure vessel with various composite materials will be conducted. Static analysis will be used in this project to find the strain, stress, and deformation. Using EN 32 Steel, Carbon fibre, and E-glass fibre materials, fatigue analysis is used to calculate the pressure vessel's life, damage, and safety factor.
Using EN 32 Steel, Carbon fibre, and E-glass fibre materials, thermal analysis is used to calculate the temperature distribution and heat transfer rate per unit area of the pressure vessel. Finding the stress, deformation, and strain at various layer stackings, such as layers 3, 6, 9, and 12, is the purpose of linear layer analysis.

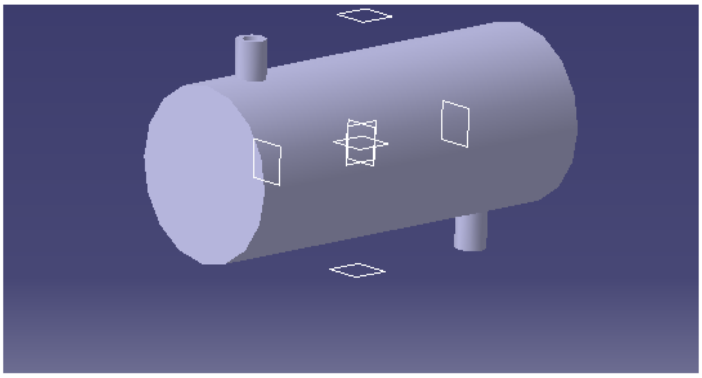
2. Body of Paper

Dimensions
Parameters Values
Heat exchanger length= 495mm
Inner diameter of shell =200mm
Outer diameter of tube =23 mm
Inner diameter of Tube =19 mm
Number of tubes =5



3D model (shaded

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 08 Issue: 01 | January - 2024 SJIF Rating: 8.176 ISSN: 2582-3930



surface model for CFD analysis

IV METHODOLOGY:

In all of these approaches the same basic procedure is followed. During pre-processing, the geometry (physical bounds) of the problem is defined. The volume occupied by the fluid is divided into discrete cells (the mesh). The mesh may be uniform or non-uniform. The physical modelling is defined – for example, the equations of motion + enthalpy + radiation + species conservation Boundary conditions are defined. This involves specifying the fluid behaviour and properties at the boundaries of the problem. For transient problems, the initial conditions are also defined. The simulation is started and the equations are solved iteratively as a steady-state or transient. Finally a postprocessor is used for the analysis and visualization of the resulting solution

CFD ANALYSIS OF SHELL AND TUBE HEAT EXCHANGER

→→Ansys → workbench→ select analysis system → fluid flow fluent → double click →→Select geometry → right click → import geometry → select browse →open part → ok→→ select mesh on work bench → right click →edit → select mesh on left side part tree → right click → generate mesh → Select faces → right click → create named section → enter name → water inlet .Select faces → right click → create named section → enter name → water outlet

Model → energy equation → on.

Viscous → edit → k- epsilon

Enhanced Wall Treatment → ok

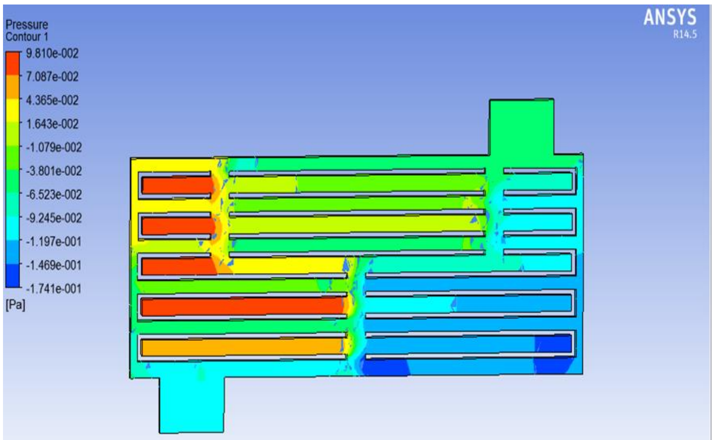
Materials → new → create or edit → specify fluid material or specify properties → ok
Select air and water

Boundary conditions → select water inlet → Edit → Enter fluid Flow Rate → 0.065Kg/s and Inlet Hot fluid Temperature –1800C and cold fluid-280C

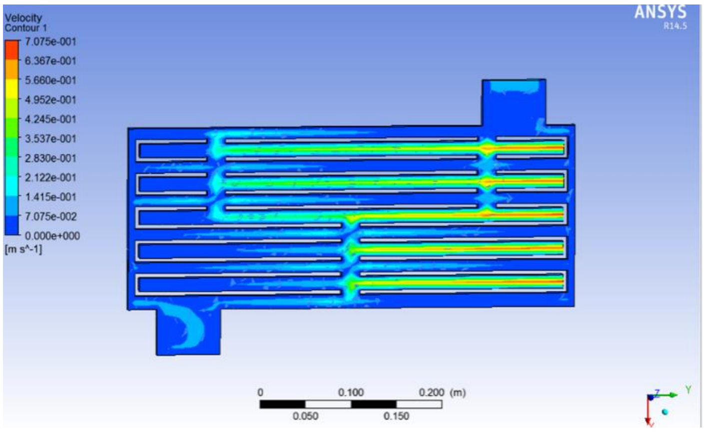
Solution → Solution Initialization → Hybrid Initialization →done

Run calculations → no of iterations = 50 → calculate → calculation complete

→→ Results → graphics and animations → contours → setup

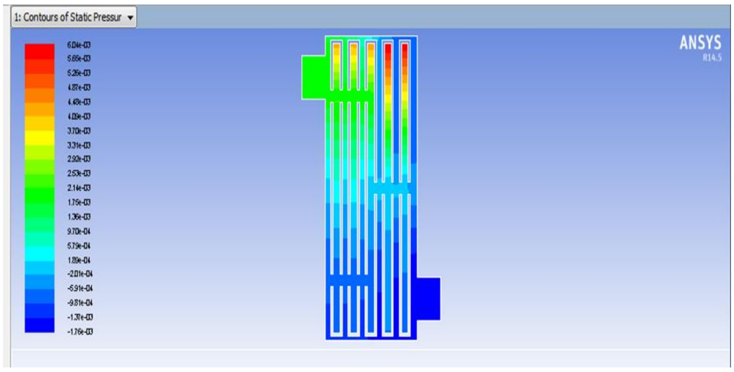


PRESSURE



PRESSURE

VELOCITY

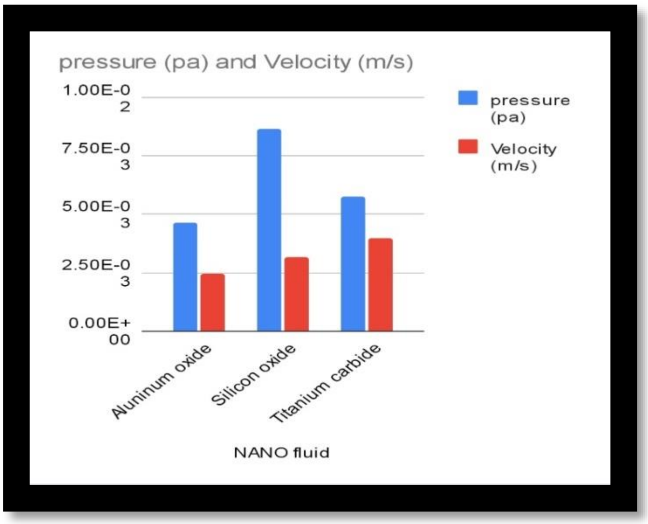


International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 08 Issue: 01 | January - 2024 SJIF Rating: 8.176 ISSN: 2582-3930

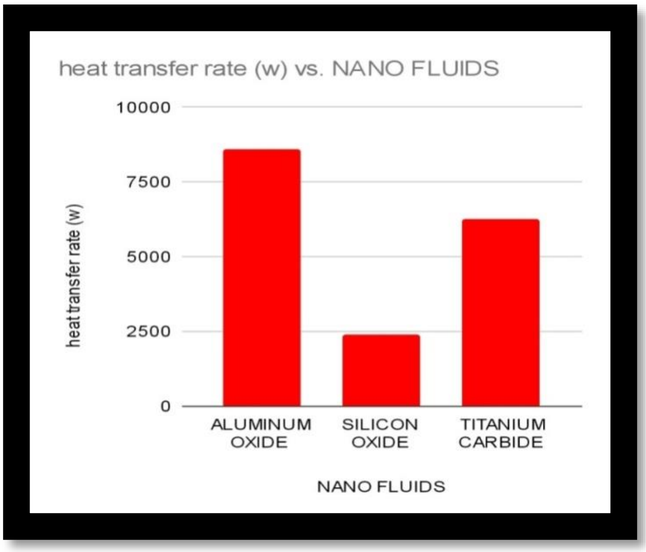
RESULT TABLES CFD ANALYSIS RESULTS

```
Total Heat Transfer Rate                 (w)
------------------------- --------------------
                      c_i            42092.875
                      c_o                    0
                      h_i            51.059849
                      h_o                    0
           wall-split.5_                     0
         ---------------- --------------------
                      Net            42143.935
```

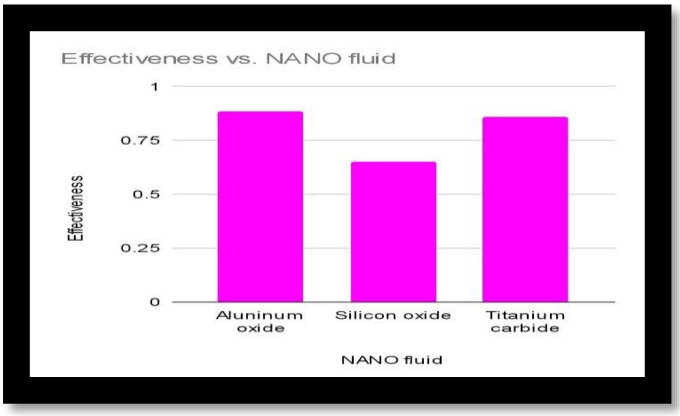
NANO FLUIDS		PRESSURE N/MM^2	VELOCITY M/SEC	TEMP (K)	HEAT TRANSFER RATE
Aluminum oxide	**0.7**	5.778e-003	7.07e-001	6.70e+03	12512.75
	0.8	5.761e-003	2.874e-004	8.61e+03	13598.631
Silicon oxide	**0.7**	8.088e-03	2.624e-004	5.41e+02	21295.345
	0.8	8.203e-03	4.023e-04	5.97e+02	19386.564
Titanium carbide	**0.7**	4.64e-03	3.736e-04	1.62e+04	12185.095
	0.8	4.592e-03	2.311e-004	1.39e+04	10256.476
air		9.81e-002	2.091e-004	1.06e+01	10142.275
water		6.042e-03	8.682e-04	6.04e+03	41432.925



PRESSURE AND VELOCITY WITH DIFFERENT NANO FLUIDS



HEAT TRANSER RATE AND NANO FLUIDS



GERAPH EFFECTIVENESS AND NANO FLUID

CONCLUSION

1) The velocity distribution of the aluminum oxide is less compare to silicon oxide and titanium carbide. The velocity distribution aluminum oxide which gives better performance.

2. The heat transfer rate is more in aluminum oxide because viscosity is less compare to silicon oxide, titanium carbide. In aluminum oxide the Nano particles which are mixed with same concentration ratio then viscosity range increase then the fluid moves slow then heat transfer rate is more in aluminum oxide.

3. Effectiveness is more for aluminum oxide compare to silicon oxide and titanium carbide

4. By testing these three nano fluids the aluminum oxide which gives better performance compare to silicon oxide and titanium carbide

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. M. VishnuPrasad, Raja Sekhara Rao Mettu, Dr. J. Pavanu Sai "DESIGN AND MICROSTRUCTURE ANALYSISOF TIG WELDING WITH TWO DIS-SIMILAR MATERIALS " 2024 IJCRT | Volume 12, Issue 1 January 2024 | ISSN: 2320-2882

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

4. Ch N V Jaswanth , M VishnuPrasad * , Dr. C.H V. S. Parameswara Rao, Dr J. Pavanu Sai "Design and Analysis of Multilayered Pressure Vessel" International Journal of Research Publication and Reviews, Vol 5, no 1, pp 1440-1446 January 2024 ISSN 2582-7421.

5.Rajagapal THUNDIL KARUPPA RAJ and Srikanth GANNE, "Shell side numericalanalysis 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

6. Ch N V Jaswanth , Dr J.Pavanu Sai ,M.VishnuPrasad* "Design and Analysis of Radial Arm used in Drilling Machine". International Journal of Research Publication and Reviews, Vol 5, no 1, pp 1447-1451 January 2024, ISSN 2582-7421

7.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).

8.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.

9. Dr J. Pavanu Sai, Dr Siva Naga Malleswara Rao Singu, M.VishnuPrasad, "DESIGN AND ANALYSIS OF TURBO CHARGER TURBINE WHEEL USING COMPOSITE MATERIALS" 2024 IJRAR January 2024, Volume 11, Issue 1, (E-ISSN 2348-1269, P- ISSN 2349-5138).

10. M.Vishnu Prasad, Dr. Siva Naga Malleswara Rao Singu , Dr. J. Pavanu Sai, S. Durga Prasad, International Journal of Advanced Research in Science, Communication and Technology (IJARSCT) , Volume 4, Issue 1, January 2024 ISSN (Online) 2581-9429

11. M. Vishnu Prasad , Dr. C.H.V.S Parameswara Rao, S. Dugra Prasad , Dr. J. Pavanu Sai , INTERNATIONAL JOURNAL OF PROGRESSIVE RESEARCH IN ENGINEERING MANAGEMENT AND SCIENCE (IJPREMS) Vol. 04, Issue 01, January 2024, e-ISSN :2583-1062.

12. D.Srimaniram, CH.Aslesha, Vishnuprasad mahanti and P.Surya4 , "FABRICATION AND ANALYSIS OF NATURAL COMPOSITES BY USING COCONT FIBER, RICE HULLS & SAW DUST AS REINFORCEMENT" VOLUME 9, ISSUE 6, 2022, GIS SCIENCE JOURNAL, ISSN NO : 1869-9391

13 .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.

14.M. Naveen kumar, Vishnuprasad mahant, CH.Aslesha, M.Roja, "EVOLUTION OF WELDING MICROSTRUCTURE USE SIMILAR & DISSIMILAR WELD JOINTS BETWEEN DIFFERENT ANGLES" VOLUME 9, ISSUE 6, 2022, GIS SCIENCE JOURNAL, ISSN NO : 1869-9391.

15. Donald Q.Kern. 1965. Process Heat transfer (23rdprinting 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.