iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH
IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 5 Issue: VII Month of publication: July 2017

DOI:

CFD Analysis on Heat Transfer Through Different Extended Surfaces

Ravindra Kondaguli[1]
[1]*Department of Mechanical Engineering BLDECET Vijayapur*

Abstract: ***The present work includes CFD analysis and comparison of heat transfer analysis and pressure loss for different shape fins with Rectangular Duct when surface area is same for all. There are two shape fin are using for analysis as rectangular fin, cylindrical (circular). The purpose of this study is to determine the optimum dimensions and shapes for rectangular longitudinal fins, cylindrical pin fins by including transverse heat conduction. This analysis completed to calculated Maximum Heat transfer Rate of fin Surface and Minimum Pressure loss in Duct due to shape change. For analysis a three dimensional finite volume based CFD Tool ANSYS 15.0 Fluent was used. Model has three basic parts as solid base, solid fin surface and rectangular duct. Heat supplied to solid fin and it conducted to solid fin surface and simultaneously it is convected to air which was flowing in the duct. Models are generated in Solid Works Software. Thereafter it imported into ANSYS 15.0 Fluent. Boundary conditions were defining with appropriate material property in Fluent software. In the solver all flows were specified as steady state and incompressible. The realizable k-e turbulence model with standard wall function was set for each model for turbulent flow. The Segregated 3D solver with an implicit formulation was set to solve the models. Different results calculated at different Reynold's number for laminar flow and Turbulent flow. After solving, Post processing is completed and found different results as contour plots, X-Y Plots and Vector Plots for Laminar and turbulent flow including Heat transfer rate and pressure loss. According to result discussion on the basis of printed data it is concluded that the Rate of heat transfer is minimum for rectangular shape fin surface and maximum for Circular pin fin surface and pressure loss is minimum in duct in case of Circular fin so it is better to use where maximum heat transfer rate is required. This study also shows that numerical models backed with experimental analysis can reduce both the time and money required to create and evaluate engineering concepts, especially those that deal with fluid flow and heat transfer.***

Keywords: CFD, Heat transfer, Fins, Extended Surfaces, ANSYS FLUENT

I. INTRODUCTION

The most effective heat transfer enhancement can be achieved by using fins as elements for the heat transfer surface area extension. In the past a large variety of fins have been applied for these purposes, leading a very compact heat exchangers with only gas or gas and liquid as the working media. Plate fin rotary regenerators and tube fin are widely encountered compact heat exchangers across the industry.

The below figure shows circular and rectangular fins

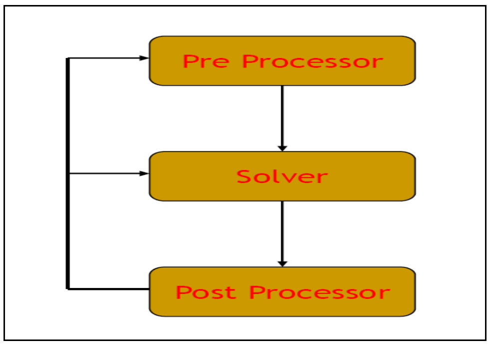
II. METHODOLOGY

The present work includes CFD analysis and comparison of heat transfer analysis and pressure loss for different shape fins with Rectangular Duct when surface area is same for all. There are two shape fin are using for analysis as rectangular fin, cylindrical (circular). The purpose of this study is to determine the optimum dimensions and shapes for rectangular longitudinal fins, cylindrical pin fins by including transverse heat conduction. This analysis completed to calculated Maximum Heat transfer Rate of fin Surface and Minimum Pressure loss in Duct due to shape change. For analysis a three dimensional finite volume based CFD Tool ANSYS 15.0 Fluent was used. Model has three basic parts as solid base, solid fin surface and rectangular duct. Heat supplied to solid fin and it conducted to solid fin surface and simultaneously it is convected to air which was flowing in the duct. Models are generated in Solid Works Software. Thereafter it imported into ANSYS 15.0 Fluent. Boundary conditions were defining with appropriate material property in Fluent software. In the solver all flows were specified as steady state and incompressible. The realizable k-e turbulence model with standard wall function was set for each model for turbulent flow. The Segregated 3D solver with an implicit formulation was set to solve the models. Different results calculated at different Reynold's number for laminar flow and Turbulent flow. After solving, Post processing is completed and found different results as contour plots, X-Y Plots and Vector Plots for Laminar and turbulent flow including Heat transfer rate and pressure loss. According to result discussion on the basis of printed data it is concluded that the Rate of heat transfer is minimum for rectangular shape fin surface and maximum for Circular pin fin surface

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor:6.887
Volume 5 Issue VIII, July 2017- Available at www.ijraset.com

and pressure loss is minimum in duct in case of Circular fin so it is better to use where maximum heat transfer rate is required. This study also shows that numerical models backed with experimental analysis can reduce both the time and money required to create and evaluate engineering concepts, especially those that deal with fluid flow and heat transfer. A CFD analysis includes the following steps

A. Pre processor
B. Geometry generation
C. Geometry cleanup
D. Meshing
E. Solver
F. Problem specification
G. Additional models
H. Numerical computation
I. Post Processor
J. Line and Contour data
K. Average Values
L. Report Generation
M. Post processer



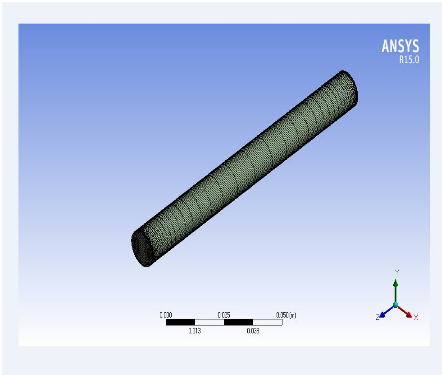
The present work consists of a heated surface to it fin is attached. Modeling is done in CATIA. Meshing is done in Hypermesh and the meshed file is imported in ANSYS 15.0.Anslysis is done in ANSYS-FLUENT.
Specification of circular fin
1) Duct size :15cm×10cm
2) Diameter of the pin : 1.27cm
3) Length of the pin : 12.5cm

III. RESULTS

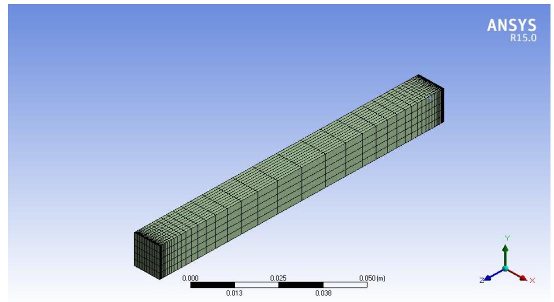
The above two tabel shows meshing of both circular and rectangular fins and meshing with fluid domain.
Mesh generation is the practice of generating a polygonal or polyhedral mesh that approximates a geometric domain. The term "grid generation" is often used interchangeably. Typical uses are for rendering to a computer screen or for physical simulation such as finite element analysis or computational fluid dynamics.

MATERIAL: BRASS	MATERIAL: COPPER

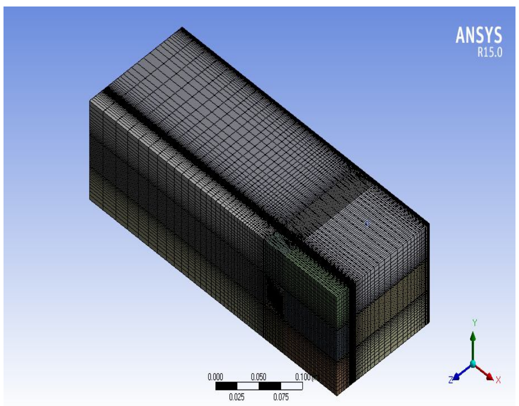
1 CIRCULAR FIN WITH MESH



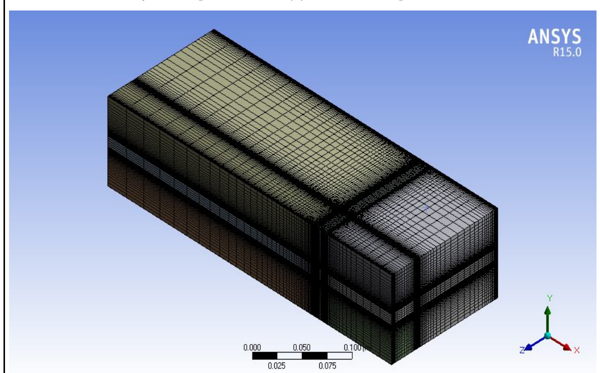
1. RECTANGULAR FIN WITH MESH



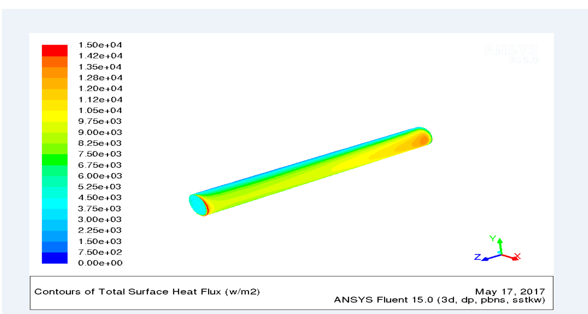
2. MESH FIN WITH FLUID



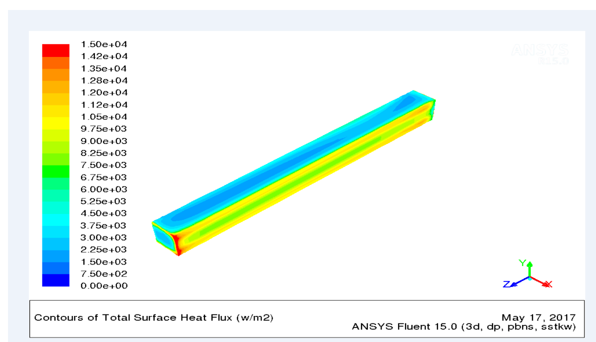
2. MESH FIN WITH FLUID



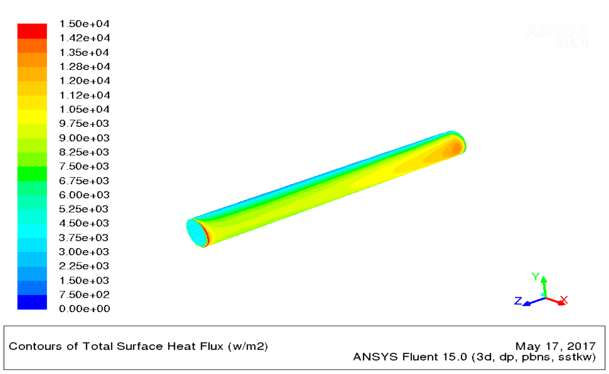
HEAT FLUX ON UP STREAM



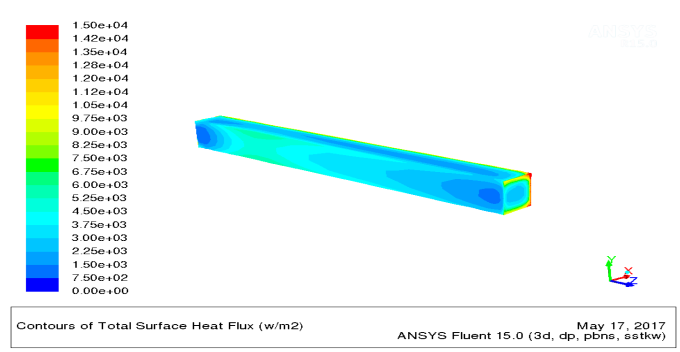
HEAT FLUX ON UP STREAM



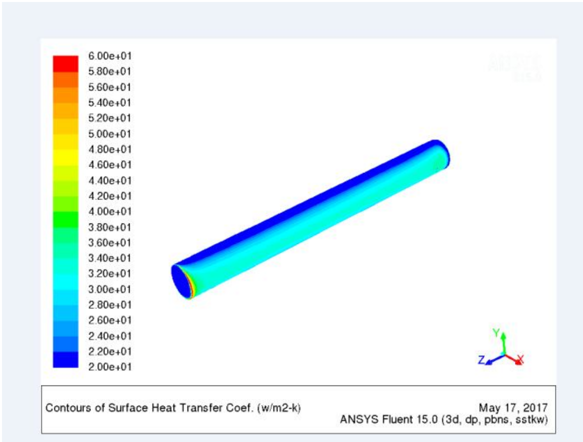
HEAT FLUX ON DOWN STREAM



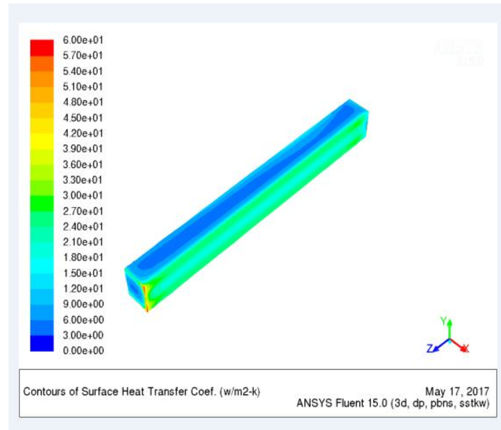
HEAT FLUX ON DOWNSTREAM



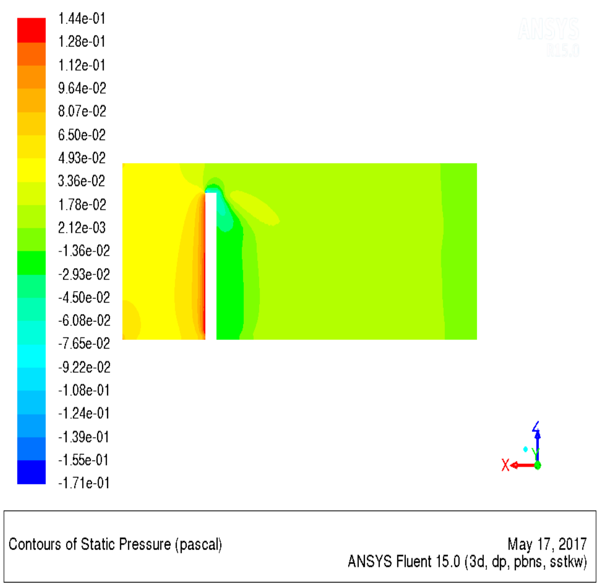
SURFACE HEAT TRANSFER COEFFICIENT (UPSTREAM)



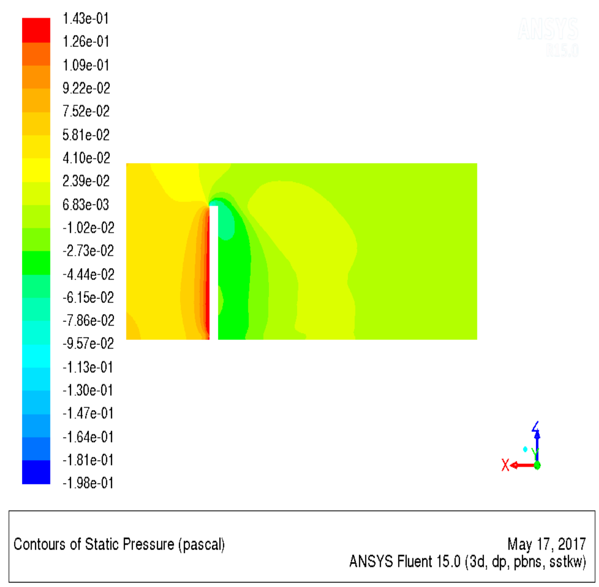
SURFACE HEAT TRANSFER COEFFICIENT (UPSTREAM)



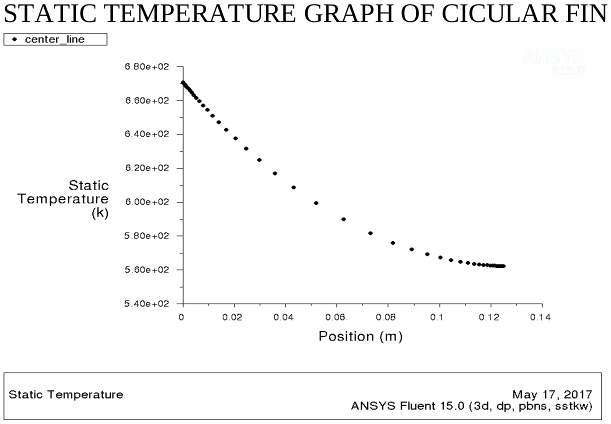
STATIC PRESSURE OF CIRCULAR FIN



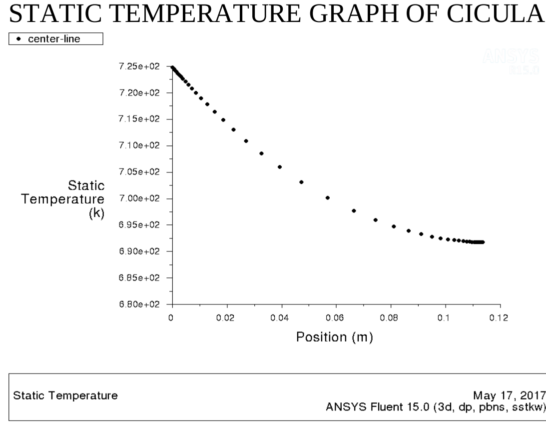
STATIC PRESSURE OF CIRCULAR FIN



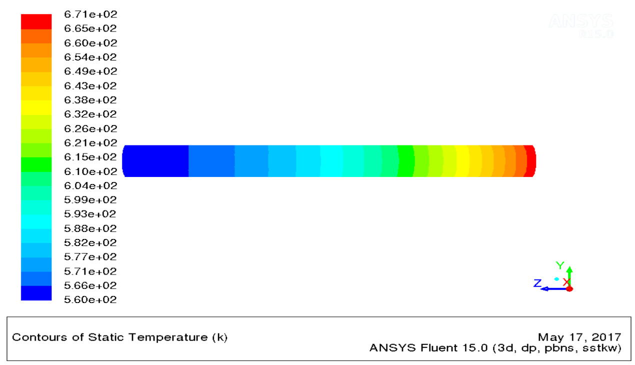
STATIC TEMPERATURE GRAPH OF CICULAR FIN



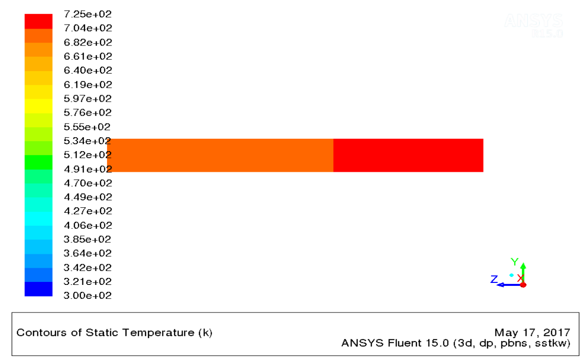
STATIC TEMPERATURE GRAPH OF CICULAR FIN



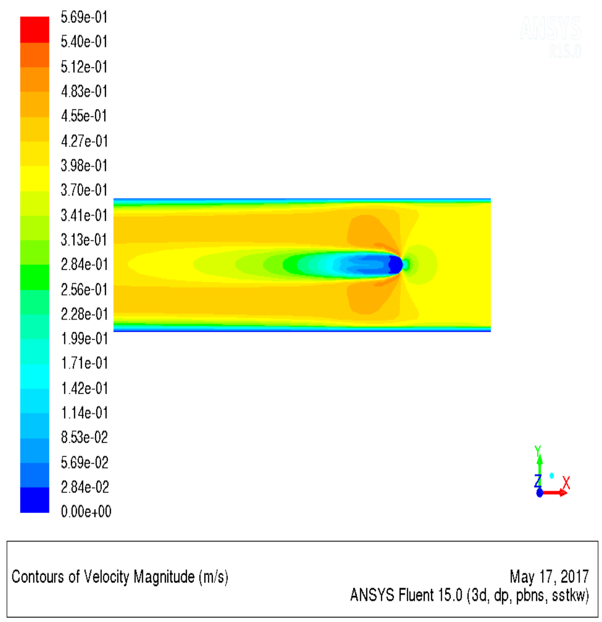
STATIC TEMPERATURE IN (K)



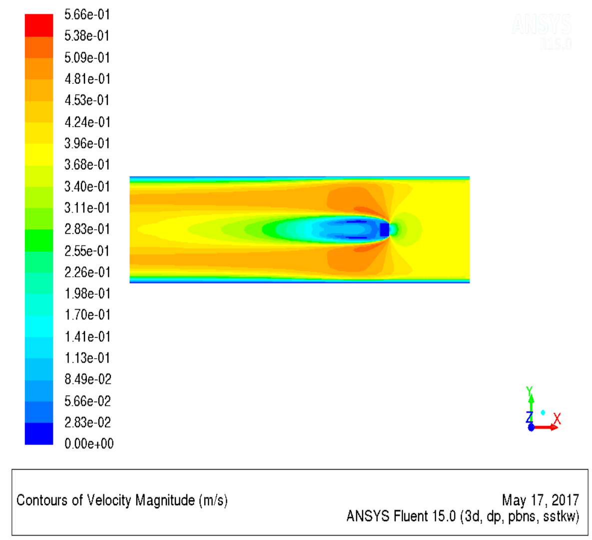
STATIC TEMPERATURE IN (K)



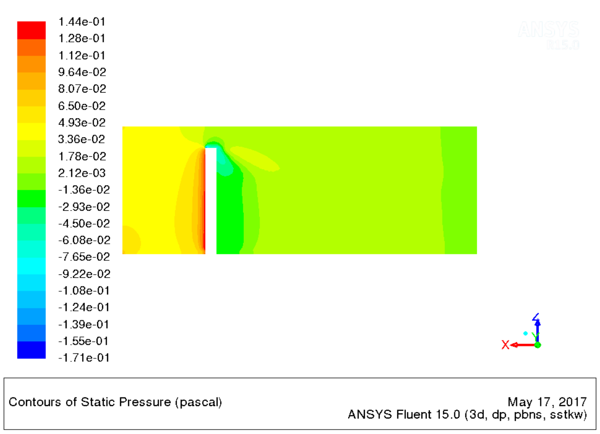
VELOCITY MAGNITUDE (m/s) OF CICULAR FIN



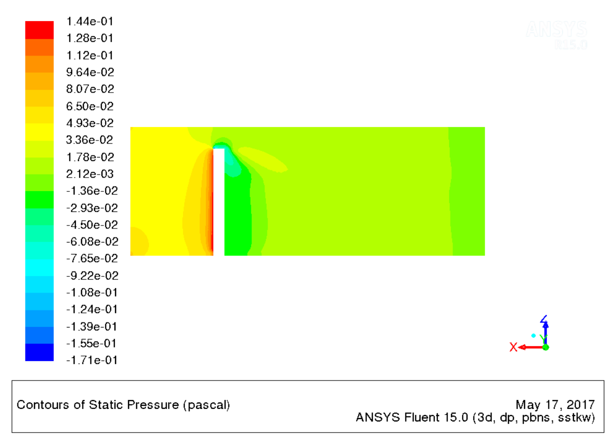
VELOCITY MAGNITUDE (m/s) OF RECTANGULAR FIN



STATIC PRESSURE OF CIRCULAR FIN



STATIC PRESSURE OF CIRCULAR FIN



IV. CONCLUSION

The static Temperature for circular fin at the tip = 560 K and for rectangular fin tip = 695 K.So the temperature is maximum at rectangular fin. Heat transfer coefficient of circular fin is =60 W/$m^2 k$ and reactangular fin =60 W/$m^2 k$. heat transfer coefficient is similar for circular fin and rectangular fin. For Laminar Flow static Pressure of circular fin is =0.144 Pascal and Rectangular fin = 0.143Pa so Pressure loss is minimum forrectangular fin and maximum forCircular fin in the Duct. In the duct Air is Flowing and it absorbs heat from fin surface. Air gets maximum heat from that fin which released (dissipated) maximum heat. Air gets Heat and increases temperature of air in the case of circular fin =671 K, Rectangular fin = 725 K (for Laminar flow).

REFERENCES

[1] Er. Mukesh didwania, (M. Tech), LIET, Alwar (Rajasthan) Dissertation on 'Study and Analysis of Heat Transfer of different shapes fins using CFD Software', 2012

[2] Numerical Study of Thermal Performance of Different Pin-Fin Morphologies" Nabati H., Mahmoudi J., 46th Conference on Simulation and Modeling (SIMS 2005), Trondheim, Norway, 2005.

[3] Optimal Pin Fin Heat Exchanger Surface for Pulp and Paper Industry" Nabati H., Mahmoudi J., the Fifth International IMACS Symposium on Mathematical Modeling (5th MATHMOD), February 8 – 10, Vienna University of Technology, Vienna, Austria,2006.

[4] Numerical Modeling of a Plane Radiator Used in a Power Transformer Cooling System" Nabati H., Mahmoudi J,Submitted to the Journal of Applied Energy for publication, 2008.

[5] Chung, B. T. F. and Iyer, J. R. (1993) 'Optimum Design of Longitudinal Rectangular Fins and Cylindrical Spines with Variable Heat Transfer Coefficient', Heat Transfer Engineering, 14:1, 31 – 42, Online Publication Date: 01 January 1993.

[6] Rizos N. Krikkis, Panagiotis Razelos, Optimum Design of Spacecraft Radiators With Longitudinal Rectangular and Triangular Fins, Journal of Heat Transfer, OCTOBER 2002, Vol. 124 Õ 805.

[7] H. Versteeg, W. Malalasekra, An Introduction to Computational Fluid Dynamics: The Finite Volume Method Approach, 3rd Ed., Prentice Hall, 1996.

[8] J. P. Hallman, A text book of Heat Transfer, TMH

[9] Computational fluid dynamics: principles and applications, J. Blazek, Alstom Power Ltd., Baden-Daettwil, Switzerland , ELSEVIER 2001

[10] John D. Anderson, Jr., University of Mayland Computationa1 Fluid Dynantics: The Basics with Applications, McGraw-Hill Series

Yunus A. Cengel , Heat Transfer 2nd edition, Prentice hall page 157-159 , 2002