iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 7 **Issue:** XII **Month of publication:** December 2019

DOI: http://doi.org/10.22214/ijraset.2019.12056

Numerical Study of Fluid Flow Behavior in a Curved Heat Exchanger Channel

Aditya Pratap Singh Rathore
Department of Aerospace Engineering, Chandigarh University, India

Abstract: In this study, a numerical analysis was performed on a heat exchanger channel. This numerical study was done to simulate and investigate the dynamics and heat transfer of a flow in a curved pipe heat exchanger. The main objective of this study was to determine how the flow properties like temperature and velocity change with respect to change in the geometry of pipe heat exchanger. Several geometrical configurations were investigated to determine the influence on heat transfer. The software used was StarCCM+.
Later a numerical study was also done using an insert of certain thickness to check the variation in thermal and fluid flow characteristics.
Keywords: Heat exchanger, CFD, numerical study, StarCCM+, Thermal simulation

I. INTRODUCTION

A heat exchanger is a system used to transfer heat between two or more fluids. Heat exchangers are used in both cooling and heating processes. The fluids may be separated by a solid wall to prevent mixing or they may be in direct contact.

The flow of Newtonian and non-Newtonian fluids in curved pipes has important applications in many industries, such as heat exchangers, turbine blade and combustion chamber cooling systems, heating systems, chemical reactors and mixing systems.

The interest on flows in curved channels comes from the centrifugal induced secondary flows which clearly improve the exchange and transfer rates of mass, heat, momentum and mixing.

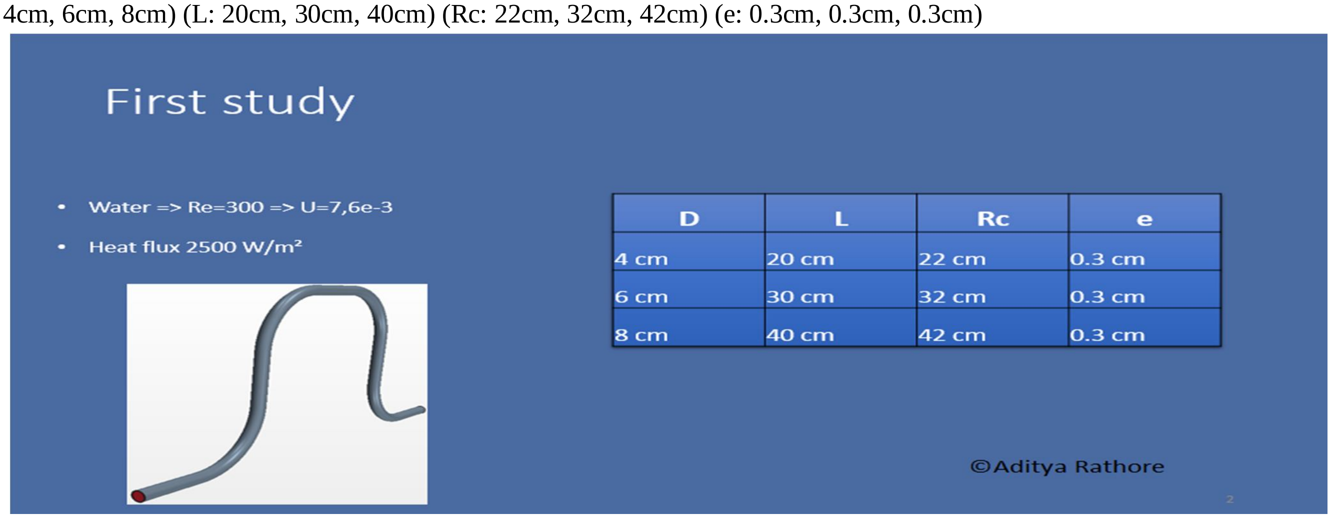
The secondary flow, in curved ducts is induced by imbalanced forces associated with centrifugal effects that create a radial pressure gradient leading to flow from the inner wall to the outer wall regions through the central part of the channel cross-section and balanced by a return flow from outer wall to the inner wall along the top and bottom channel walls.

In the laminar regime, the secondary flow in a curved channel is characterized by a pair of symmetrical vortices occupying the entire crossed by a return flow from the outer section of the channel.

In this study, heat exchanger was simulated with different geometrical parameters – diameter (D), length(L), radius of curvature(Rc) and thickness(e). Then the flow characteristics – velocity (U) and temperature (T) were compared among these geometrical parameters. The fluid medium was water with Reynolds number Re = 300 (laminar). The velocity taken was, U = 7.6e-3 m/s. The heat flux taken was 2500 W/m2. Later another pipe of thickness 1cm was inserted and simulated. Then the changes regarding temperature and velocity were investigated as well.

The set of geometrical parameters were

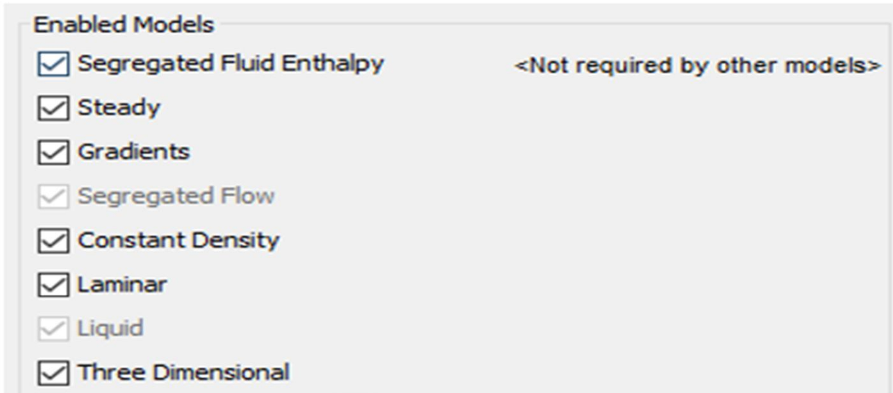
(D: 4cm, 6cm, 8cm) (L: 20cm, 30cm, 40cm) (Rc: 22cm, 32cm, 42cm) (e: 0.3cm, 0.3cm, 0.3cm)



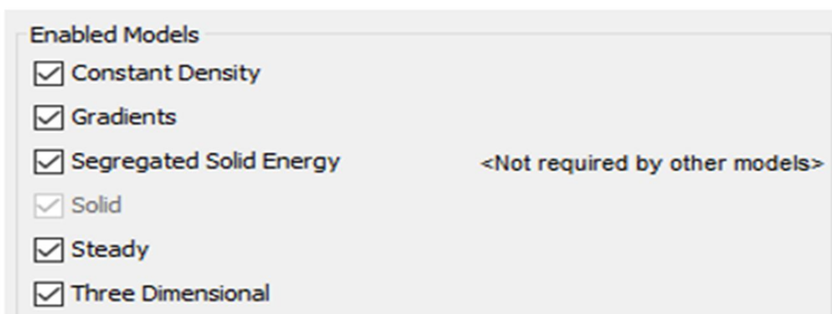
D	L	Rc	e
4 cm	20 cm	22 cm	0.3 cm
6 cm	30 cm	32 cm	0.3 cm
8 cm	40 cm	42 cm	0.3 cm

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.177
Volume 7 Issue XII, Dec 2019- Available at www.ijraset.com

II. SIMULATION PHYSICS

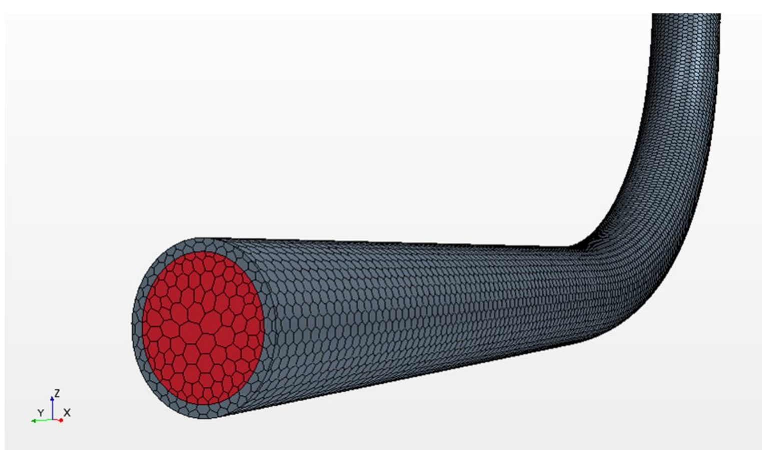


Water ©Aditya Rathore



Tube ©Aditya Rathore

III. THE MESH

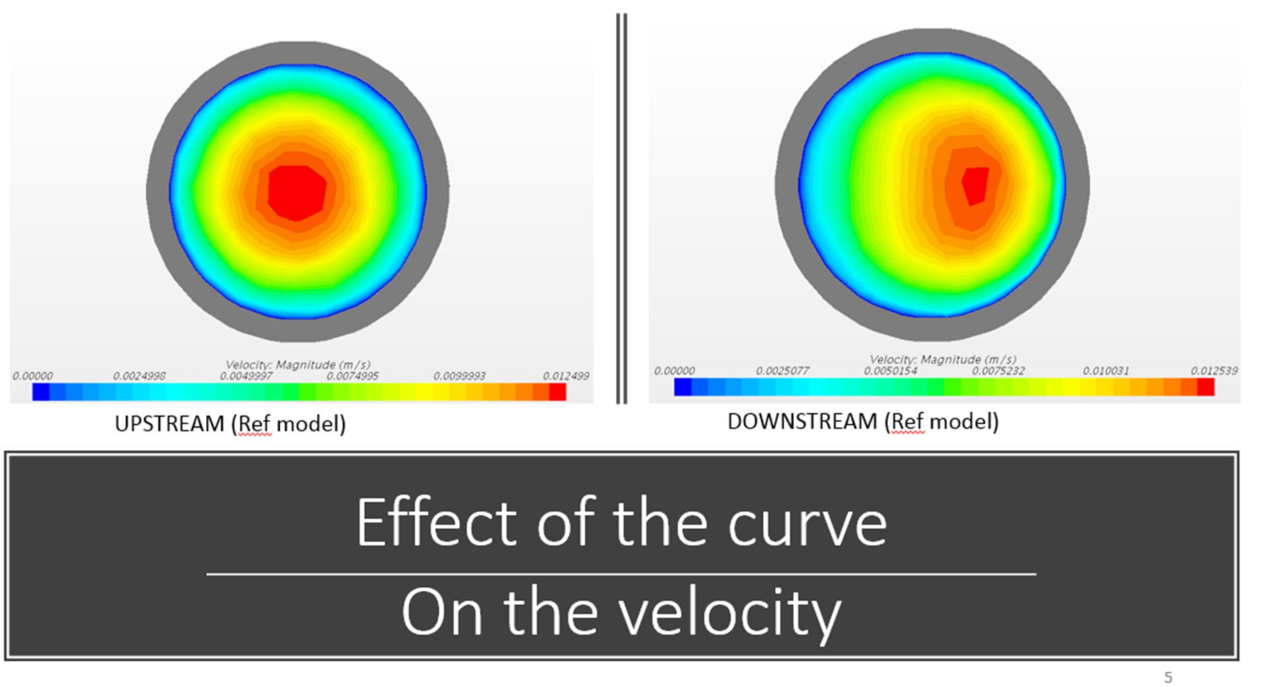


International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.177
Volume 7 Issue XII, Dec 2019- Available at www.ijraset.com

IV. RESULTS

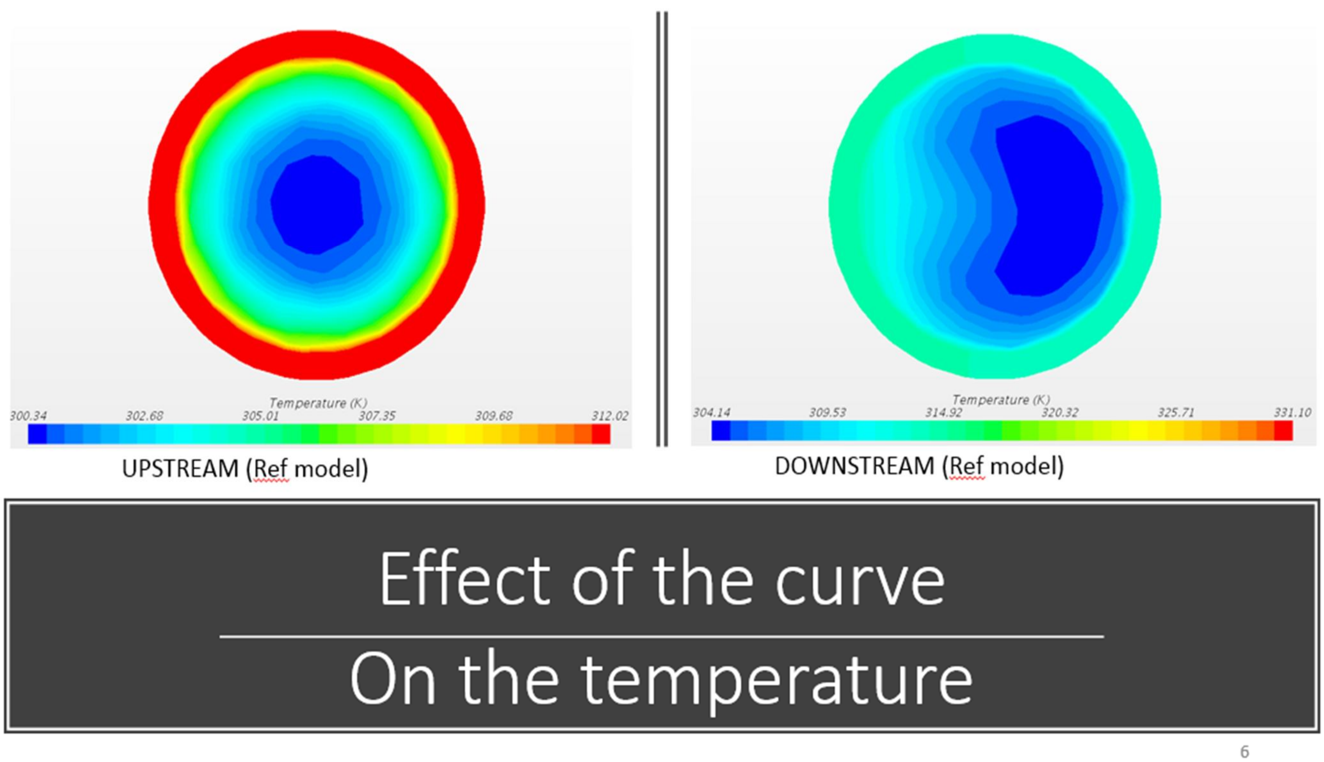
A. Velocity

Maximum velocity for upstream and downstream flow was 0.012499 m/s & 0.012539 m/s



B. Temperature

Maximum temperature for upstream and downstream flow was observed closer to the outer layer. The values were 312.02 K & 314.92 K for upstream and downstream respectively.



International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.177
Volume 7 Issue XII, Dec 2019- Available at www.ijraset.com

V. VARIATIONS

After the primary simulations were completed and the effects of the curve on velocity and temperature were observed, both for the upstream and downstream flow respectively, some geometrical variations were done namely: diameter, length, radius of curvature and thickness.

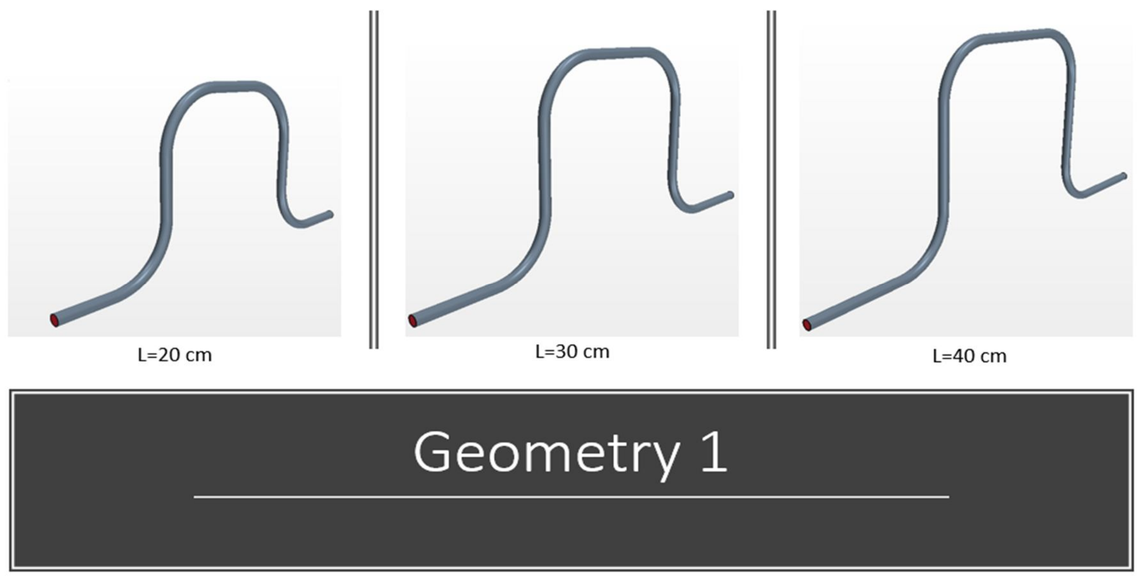
The variations were

1) Length – 20cm, 30cm, 40cm
2) Radius of curvature – 22cm, 32cm, 42cm
3) Diameter – 4cm, 6cm, 8cm
4) Wall thickness via insertion of a coaxial pipe of wall thickness 1cm.

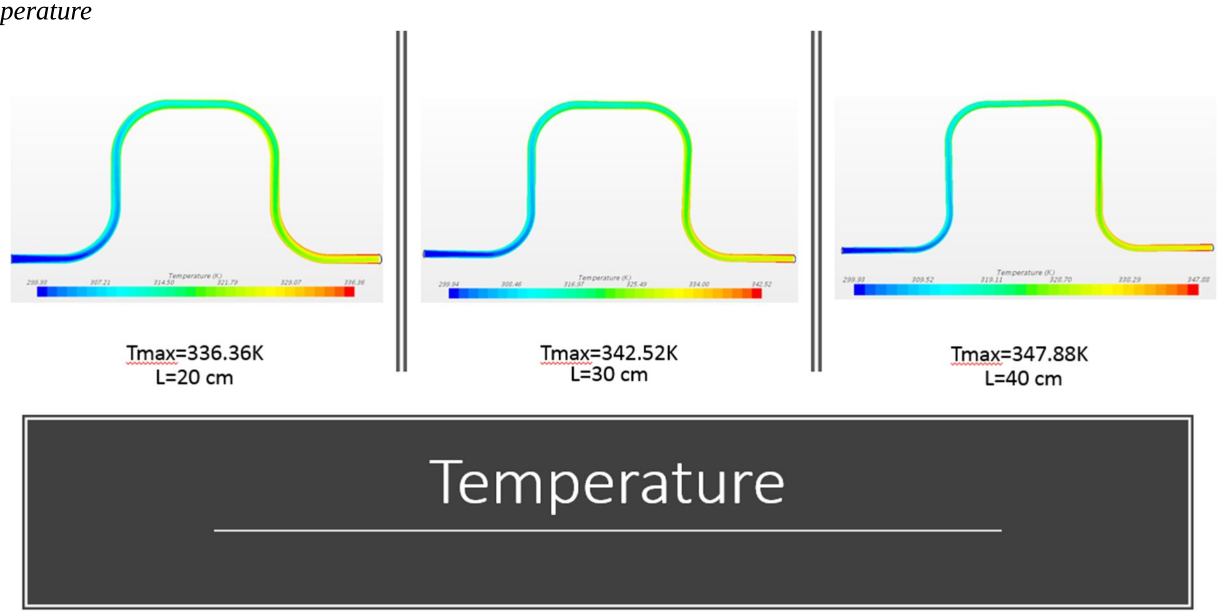
A. The Simulation Results after these Variations were Observed

The results are presented as follows.

1) *Variations with length: 20cm, 30cm and 40cm*

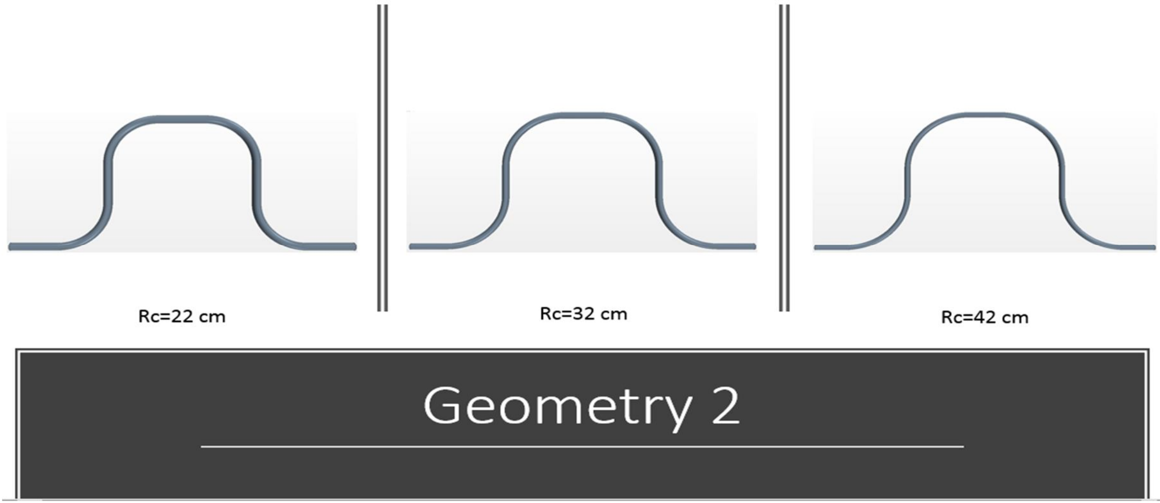


2) *Temperature*

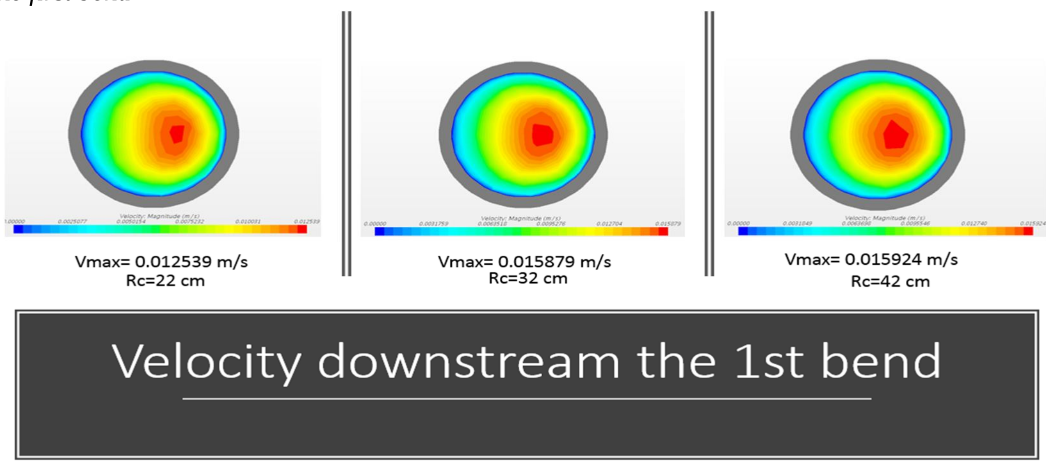


International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.177
Volume 7 Issue XII, Dec 2019- Available at www.ijraset.com

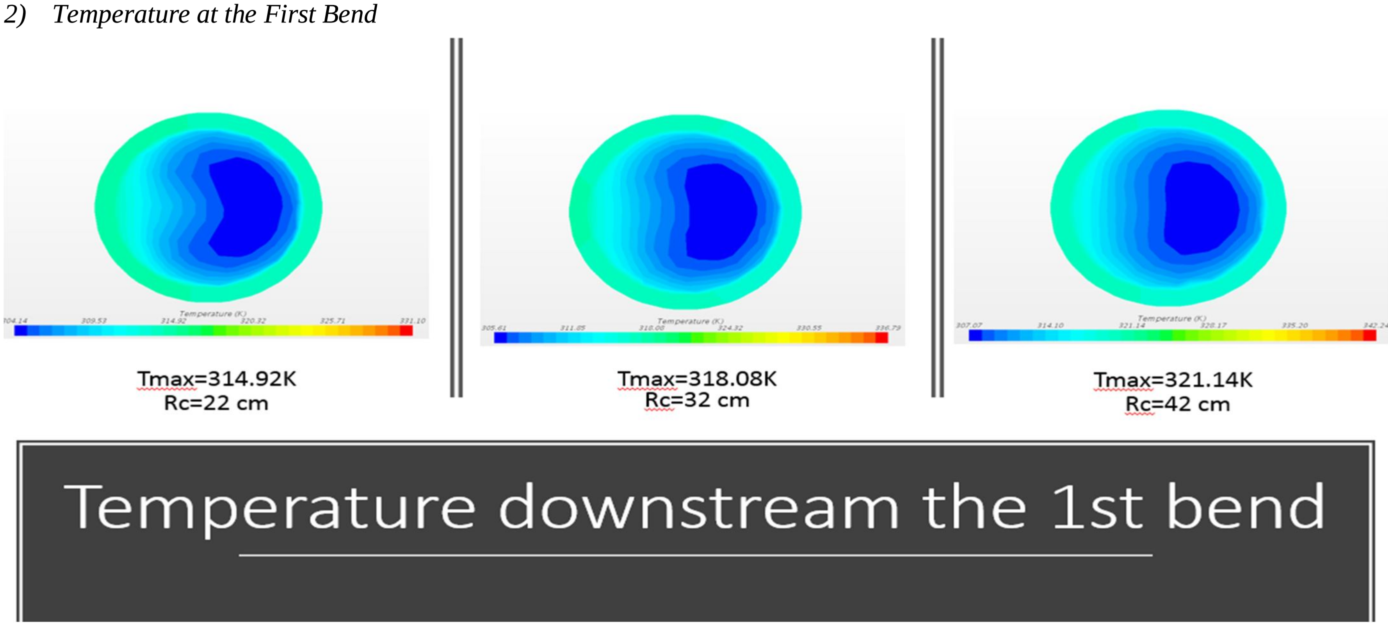
B. Variations with Radius of curvature: 22cm, 32cm and 42cm



1) Velocity at the first bend



2) Temperature at the First Bend

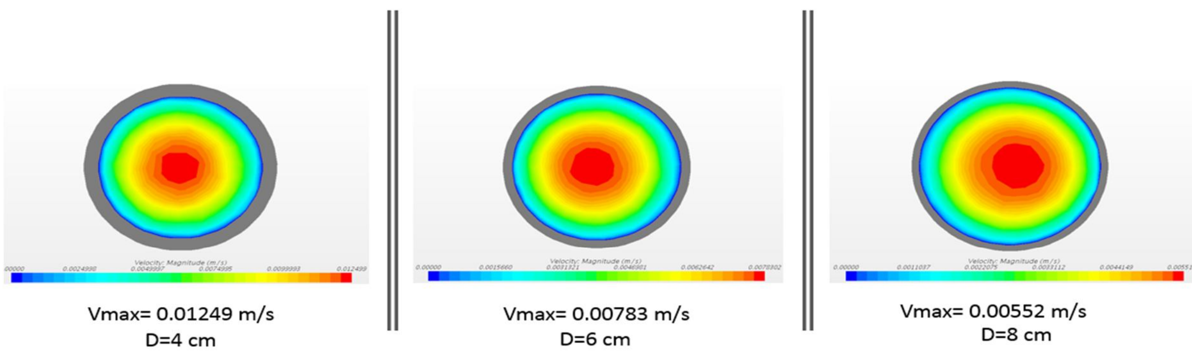


International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.177
Volume 7 Issue XII, Dec 2019- Available at www.ijraset.com

C. Variations with Diameter: 4cm, 6cm and 8cm

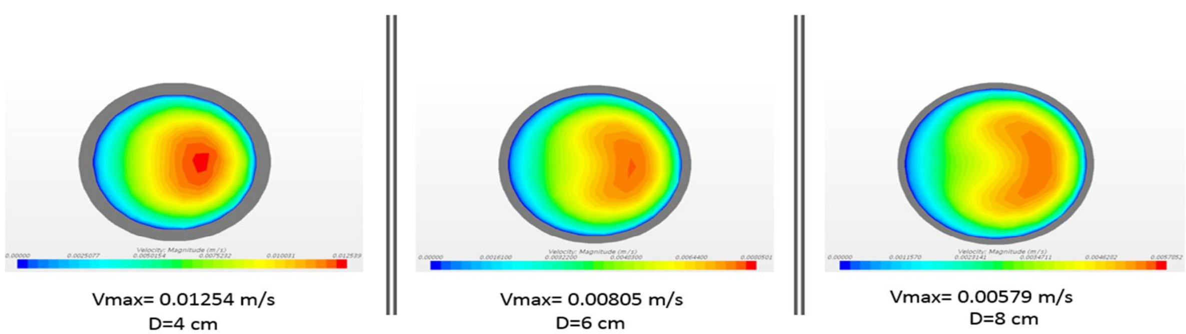
Geometry 3

1) Velocity upstream at first bend



Velocity upstream the 1st bend

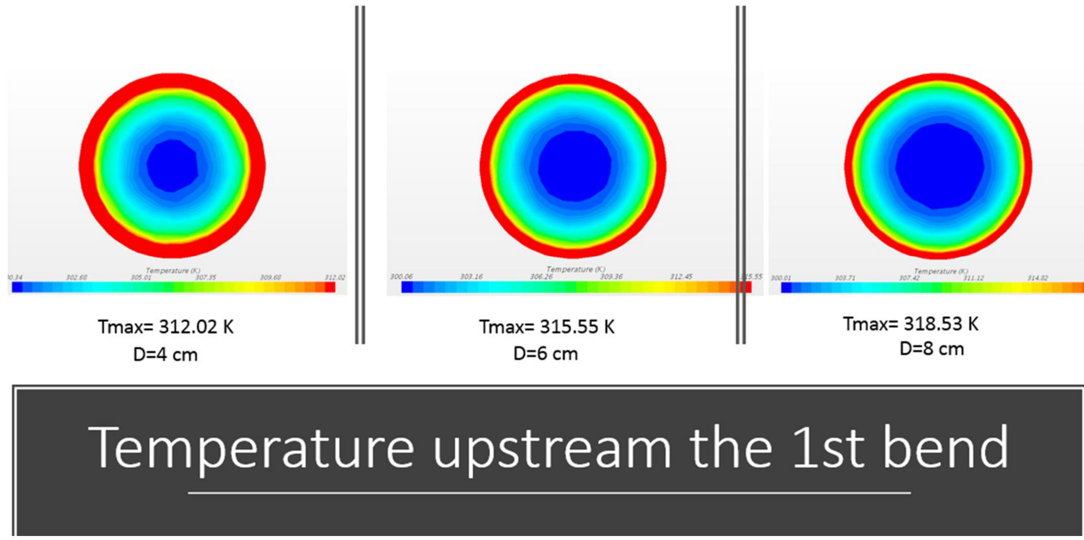
2) Velocity Downstream at first Bend



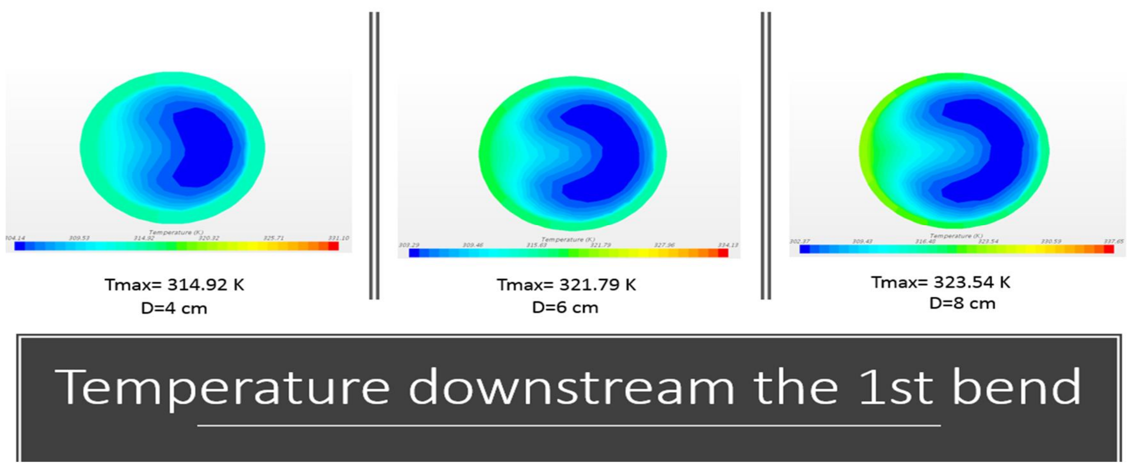
Velocity downstream the 1st bend

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.177
Volume 7 Issue XII, Dec 2019- Available at www.ijraset.com

3) Temperature Upstream at the first Bend

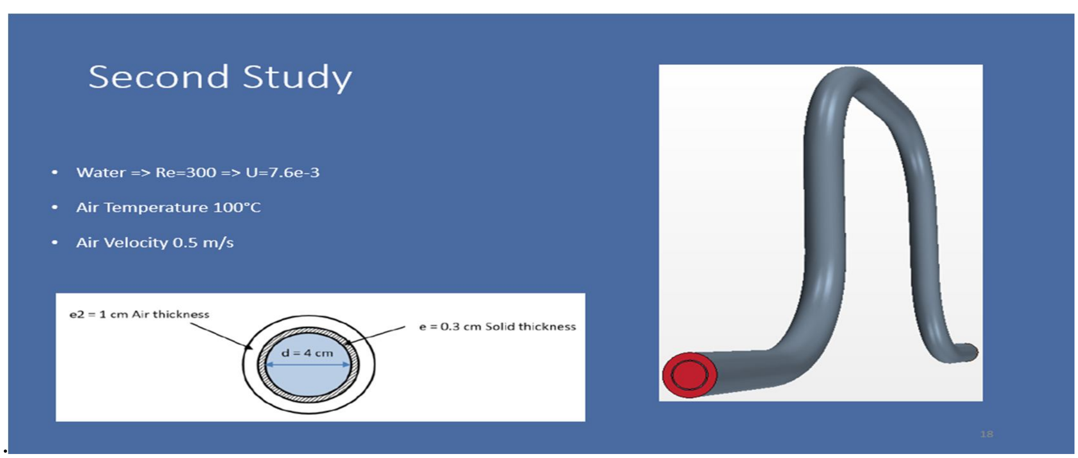


4) Temperature Downstream at the first Bend



VI. INSERTION

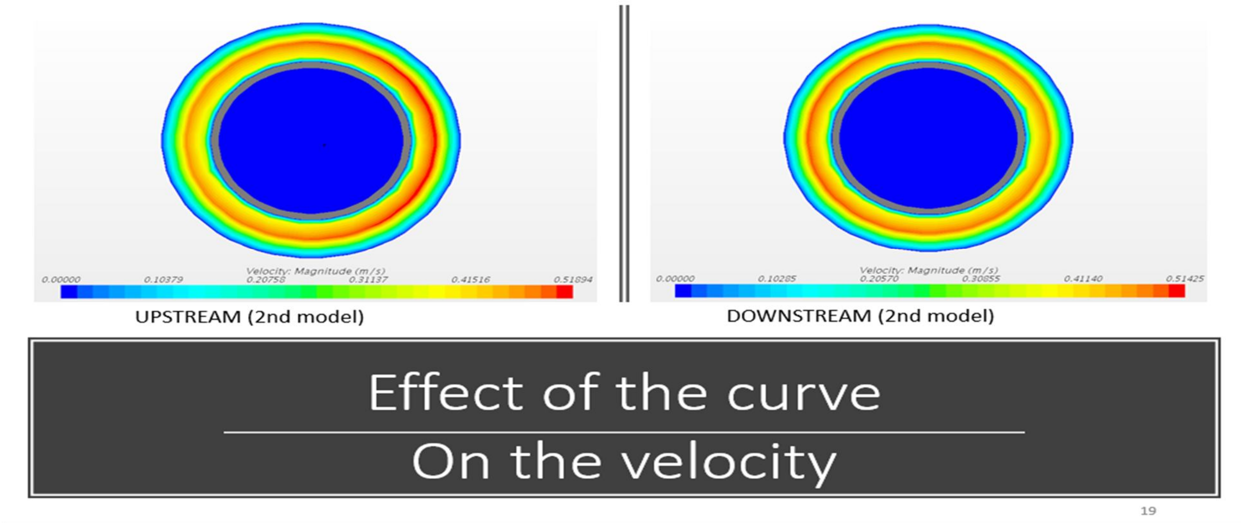
A new co-axial pipe was inserted to the initial configuration with an air-flow and a wall thickness of pipe e2 = 1cm. The temperature of the air-flow in this case was imposed as T=100 degree Celsius or 373 Kelvin and the velocity of the air, Uair = 0.5 m/s



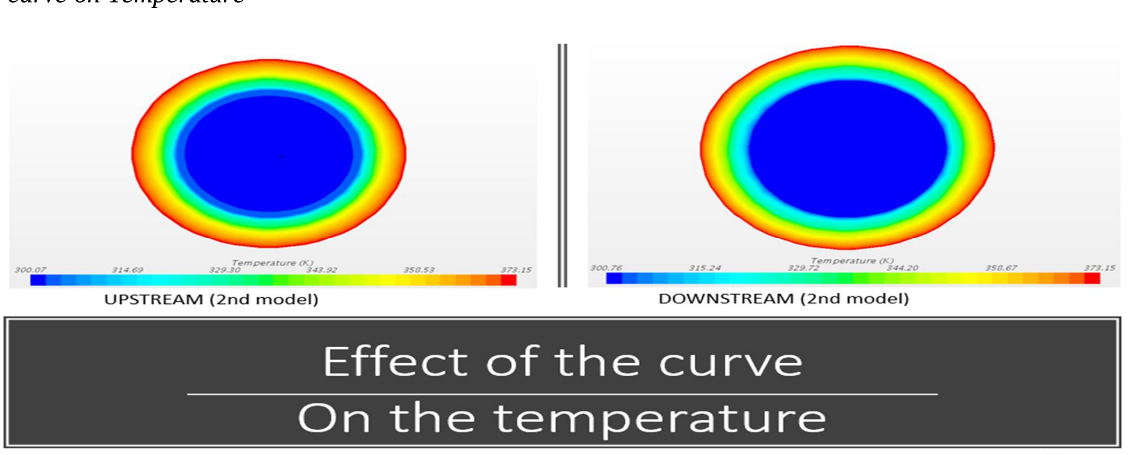
.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.177
Volume 7 Issue XII, Dec 2019- Available at www.ijraset.com

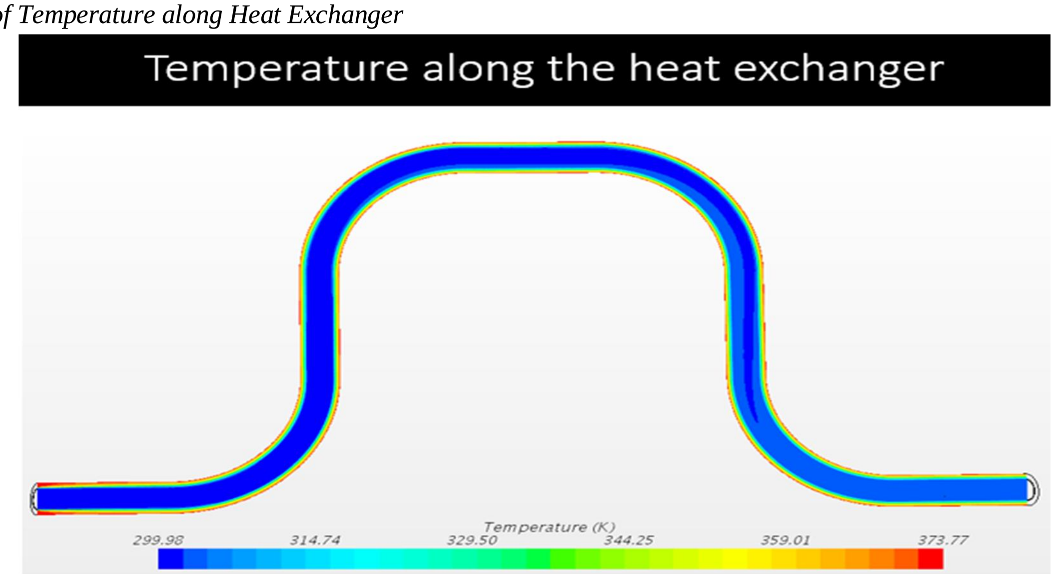
A. Effect of the Curve on Velocity



B. Effect of curve on Temperature



C. Variation of Temperature along Heat Exchanger



VII. CONCLUSION

After the numerical study and investigations, it was concluded that the flow properties of the fluid in the heat exchanger change with the change in geometrical parameters. It is evident from the simulation results that the velocity of the fluid flow decreases with the increase in diameter of the heat exchanger for both upstream and downstream direction. Velocity of the fluid flow increases with the increase in radius of curvature of the bend. Temperaturemax increases with the increase in the length of exchanger and radius of curvature as well. Likewise, increase in diameter also resulted into increment of Temperaturemax.

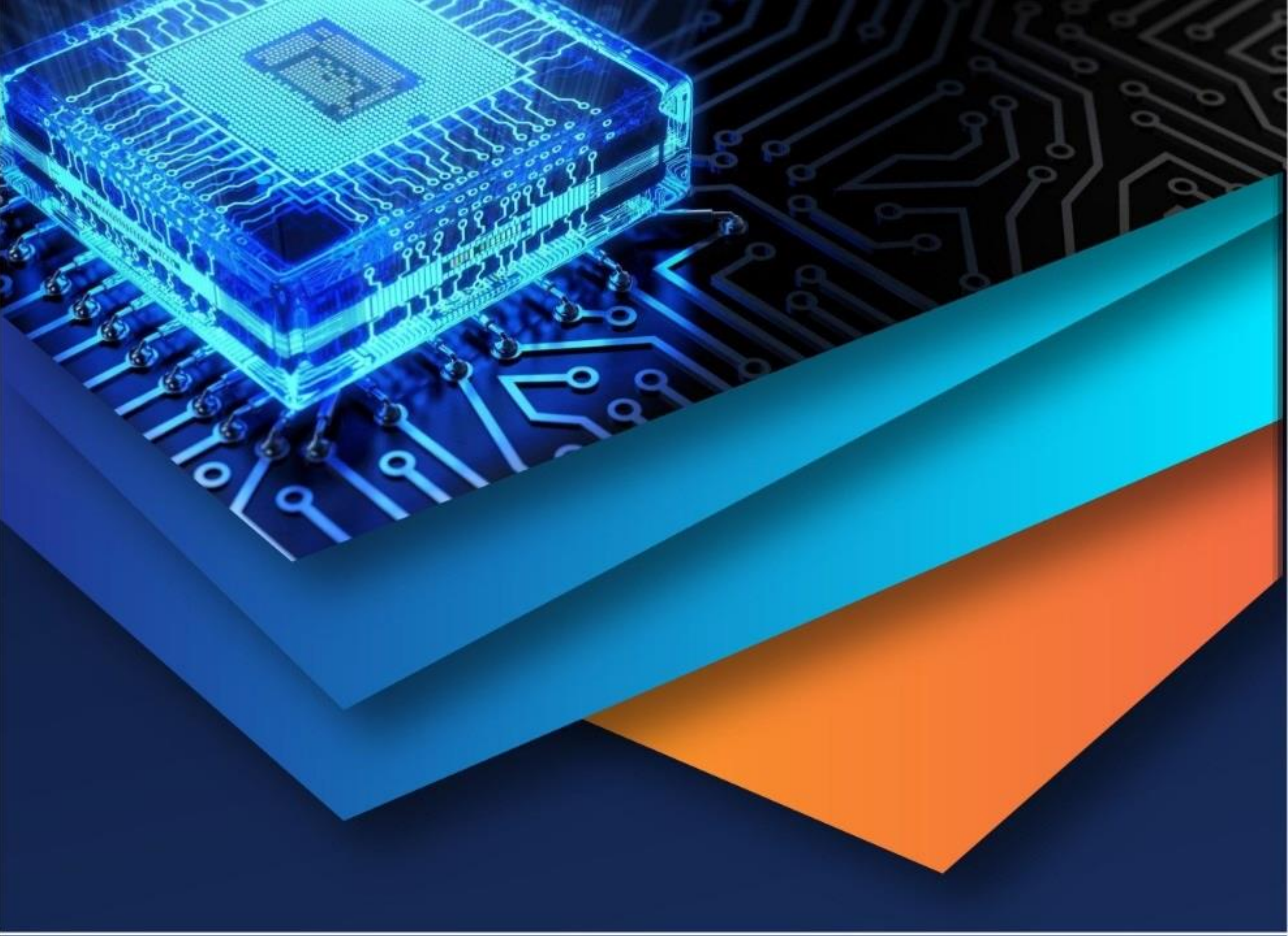
Incremented Parameter	Flow velocity	Fluid temperature (Tmax)
Diameter	Decreases	Increases
Length	No change	Increases
Radius of curvature	Increases	Increases

VIII. ACKNOWLEDGMENT

I wish to acknowledge my professors and mentors who always guided me and motivated me to do the research related activities. Special thanks to my professor Mr. Abbasi Wafik, IPSA, Paris for all the guidance and helping to tackle the problems occurred during this study.

REFERENCES

[1] Mallikarjuna, V. Seshadri and V. Raghu B. , "Numerical Analysis of Fin Side Turbulent Flow for Round and Flat Tube Heat Exchangers" , International Journal of Modern Engineering Research (IJMER), Vol. 4 , Issue 7, 2014.
[2] Piyush .V. Patil and D. G. Kumbharb, "CFD Investigation on Dimpled Fins With Parameter Variation for Heat Transfer Augmentation", International Journal of Engineering Research & Technology (IJERT), Vol. 3 Issue 6, 2014.
[3] Ahmed F. Khudheyer and Mahmoud Sh. Mahmoud, "numerical analysis of fin-tube plate heat exchanger by using CFD technique", ARPN Journal of Engineering and Applied Sciences, VOL. 6, NO. 7, JULY 2011.
[4] Firas Abd Ali Abbas, "Augmentation of Heat Transfer by Using Nanotechnology", M.Sc. Thesis, University of Al – Mustansiriya, 2014.
[5] Sinan Abdul Ghafar Ali, "Study the Effect of Upstream Riblet on Wing-Wall junction", M.Sc. Thesis, University of Tecnology, 2011.
[6] "Ansys Fluent User's Guide", Ansys Inc., South pointe 275 Technology Drive Canonsburg, 2011.



10.22214/IJRASET

INDEX COPERNICUS 45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY