DESIGN AND FLUID FLOW ANALYSIS OF HEAT EXCHANGER WITH DIFFERENT LOUVER FINS

Indala HariBabu[1], CH. Venkata Anvesh[2]

M-Tech Student (Thermal engineering), Department Of Mechanical Engineering,,SITAM Engineering College, pm palem Visakhapatnam, andrapradesh India[1]

Assistant Professor, Department Of Mechanical Engineering,SITAM Engineering College, pm palem Visakhapatnam, andrapradesh, India[2]

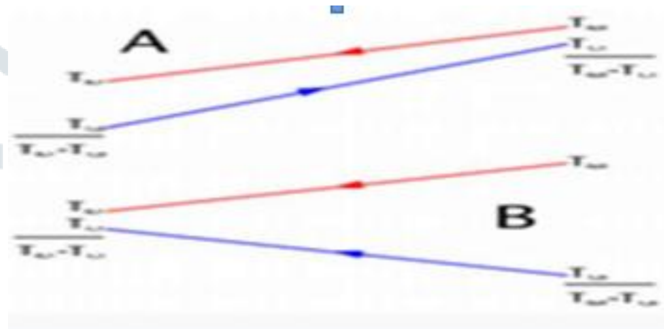
ABSTRACT:

In recent years, due to the increasing demand by industries for heat exchangers that are more efficient, compact, and less expensive, heat transfer enhancement has gained great momentum. Thus, engineers come out with one brilliant solution to deal with this issue, at which one innovation device has been developed decades ago called compact heat exchanger (CHE).In this thesis the parametric analysis on thermo-hydraulic performance of a compact heat exchanger using computational fluid dynamics (CFD). The analysis has been carried out at different frontal air velocities by varying the geometrical parameters such as louver pitch and louver angle.In this thesis the CFD analysis is to perform the heat transfer rate, mass flow rate, pressure drop and velocity at 15m/s inlet velocity..

1. INTRODUCTION

A heat exchanger is a device used to transfer heat between a solid object and a fluid, or between two or more fluids. The fluids may be separated by a solid wall to prevent mixing or they may be in direct contact. They are widely used in space heating, refrigeration, air conditioning, power stations, chemical plants, petrochemical plants, petroleum refineries, natural-gas processing, and sewage treatment. The classic example of a heat exchanger is found in an internal combustion engine in which a circulating fluid known as engine coolant flows through radiator coils and air flows past the coils, which cools the coolant and heats the incoming air. Another example is the heat sink, which is a passive heat exchanger that transfers the heat generated by an electronic or a mechanical device to a fluid medium, often air or a liquid coolant.

FLOW ARRANGEMENT

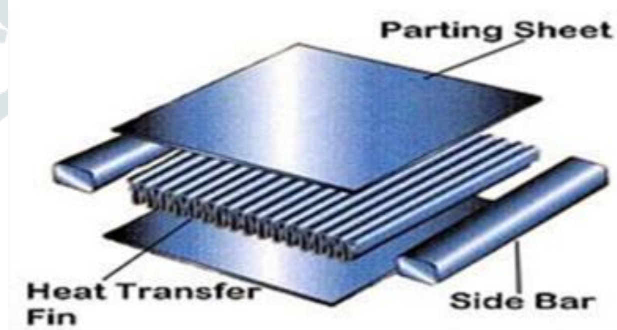


Countercurrent (A) and parallel (B) flows

Design of plate-fin heat exchangers

Originally conceived by an Italian mechanic, Paolo Fruncillo. A plate-fin heat exchanger is made of layers of corrugated sheets separated by flat metal plates, typically aluminum, to create a series of

Finned chambers. Separate hot and cold fluid streams flow through alternating layers of the heat exchanger and are enclosed at the edges by side bars.



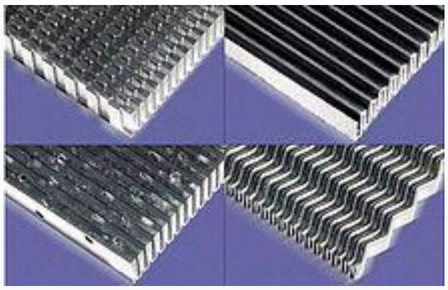
Principal Components of a Plate Fin Heat Exchanger

Heat is transferred from one stream through the fin interface to the separator plate and through the next set of fins into the adjacent fluid. The fins also serve to increase the structural integrity of the heat exchanger and allow it to withstand high pressures while providing an extended surface area for heat transfer.

A high degree of flexibility is present in plate-fin heat exchanger design as they can operate with any combination of gas, liquid, and two-phase fluids. Heat transfer between multiple process streams is also accommodated, with a variety of fin heights and types as different entry and exit points available for each stream.

Specific software is also available from various vendors, including the ProSec software for plate fin heat exchangers which was developed by ProSim in France, to simulate and design plate fin heat exchangers.

The main four types of fins are plain, which refer to simple straight-finned triangular or rectangular designs herringbone, where the fins are placed sideways to provide a zig-zag path; and serrated and perforated which refer to cuts and perforations in the fins to augment flow distribution and improve heat transfer.



Different Fin Structures for Plate Fin Heat Exchangers

A disadvantage of plate-fin heat exchangers is that they are prone to fouling due to their small flow channels. They also cannot be mechanically cleaned and require other cleaning procedures and proper filtration for operation with potentially-fouling streams.

Applications of plate-fin heat exchangers include:

- Natural gas liquefaction
- Cryogenic air separation
- Ammonia production
- Offshore processing
- Nuclear engineering
- Syngas production
- Aircraft cooling of bleed air and cabin air

Louveredfin:A **louver** (American English) or **louver** is a window blind or shutter with horizontal slats that are angled to admit light and air, but to keep out rain and direct sunshine. The angle of the slats may be adjustable, usually in blinds and windows, or fixed.

II. LITERATURE REVIEW

The Effect of Geometrical Parameters on Heat Transfer Characteristics of Compact Heat Exchanger with Louvered Fins

The present study aimed to perform the parametric analysis on thermo-hydraulic performance of a compact heat exchanger using computational fluid dynamics (CFD). The analysis has been carried out at different frontal air velocities by varying the geometrical parameters such as fin pitch, transverse tube pitch, longitudinal tube pitch, louver pitch and louver angle. The air side performance of the heat exchanger has been evaluated by calculating Colburn factor (*j*) and Fanning friction factor (*f*). The comparison of CFD results with the experimental data exhibited a good agreement and the influence of various geometrical parameters for the selected range of values on the pressure drop, heat transfer coefficient and goodness factor was analyzed. The results obtained from the analysis will be very useful to optimize the louvered fin and flat tube compact heat exchanger for better thermo-hydraulic

Performance analysis without the need of time consuming and expensive experimentation.

III. PROBLEM METHODOLOGY The analysis has been carried out at different frontal air velocities by varying the geometrical parameters such as louver pitch and louver angle.In this thesis the CFD analysis is to perform the heat transfer rate, mass flow rate, pressure drop and velocity at 15m/s inlet velocity.

INTRODUCTION TO CAD

Computer-aided design (CAD) is the use of computer

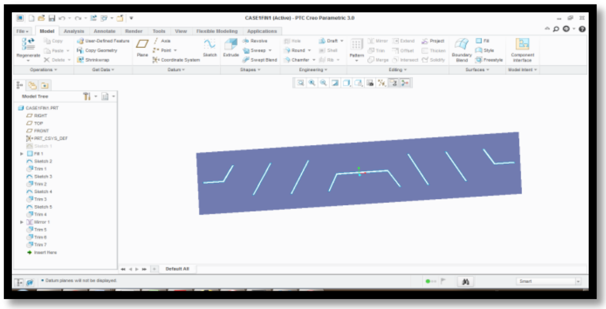
Fin pitch (mm)	Angle (^{0})
1mm	28
	30
	32
2mm	28
	30
	32
4mm	28
	30
	32

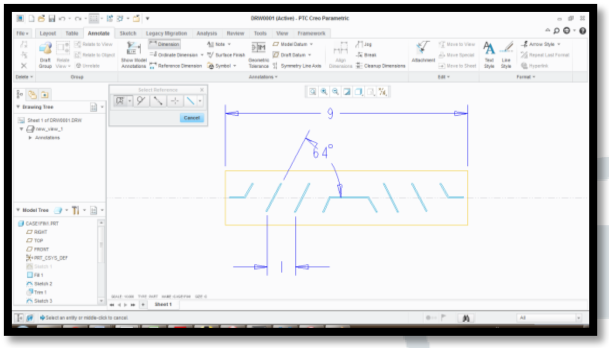
systems (or workstations) to aid in the creation, modification, analysis, or optimization of a design. CAD software is used to increase the productivity of the designer, improve the quality of design, improve communications through documentation, and to create a database for manufacturing. CAD output is often in the form of electronic files for print, machining, or other manufacturing operations. The term **CADD** (for Computer Aided Design and Drafting) is also used.

INTRODUCTION TO CREO

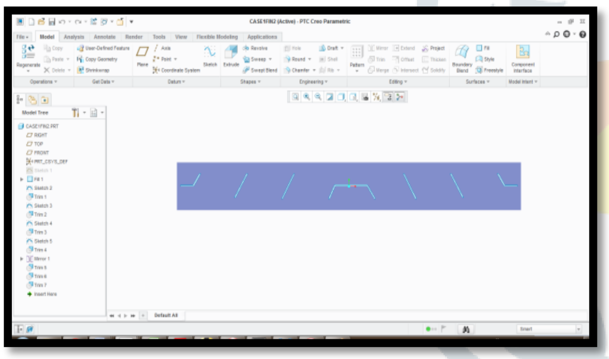
PTC CREO, formerly known as Pro/ENGINEER, is 3D modeling software used in mechanical engineering, design, manufacturing, and in CAD drafting service firms. It was one of the first 3D CAD modeling applications that used a rule-based parametric system. Using parameters, dimensions and features to capture the behavior of the product, it can optimize the development product as well as the design itself.

Case 1: fin angle 28^{0} and fin pitch 1.0mm

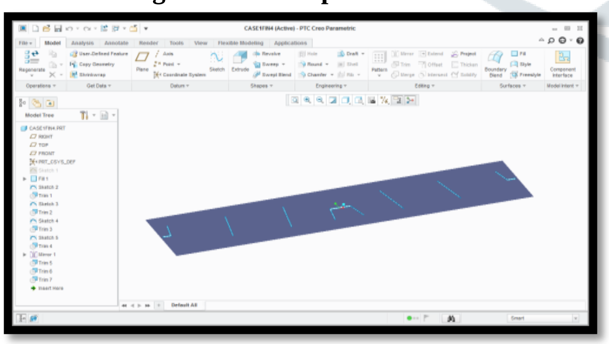




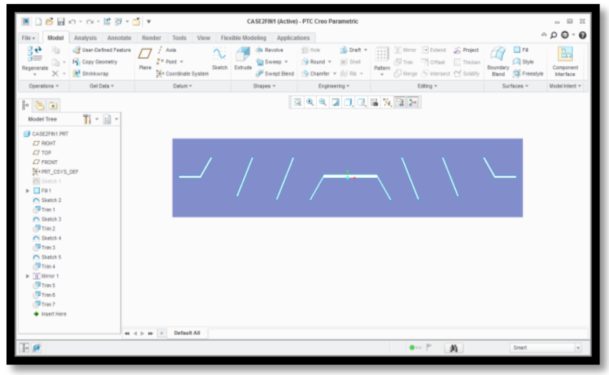
Case 1: fin angle 28^{0} and fin pitch 2.0mm



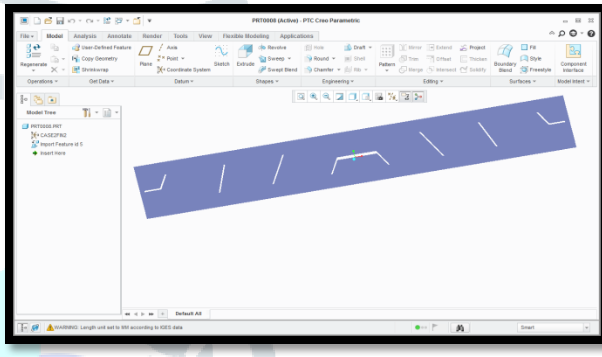
Case 1: fin angle 28^{0} and fin pitch 4.0mm



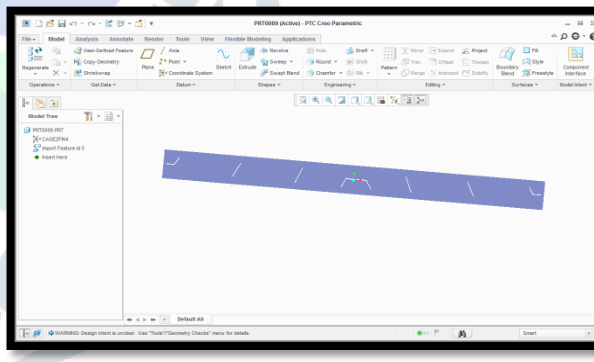
Case 2: fin angle 30^{0} and fin pitch 1.0mm



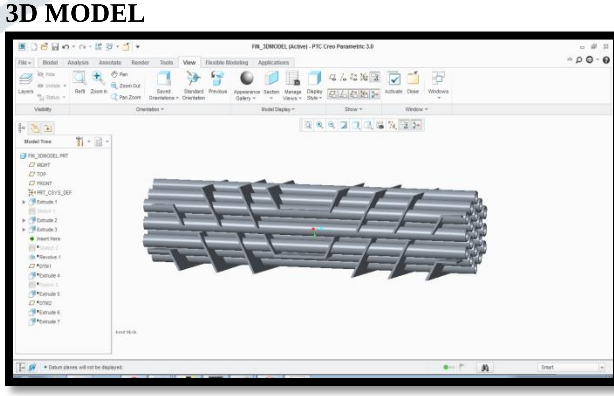
Case 2: fin angle 30^{0} and fin pitch 2.0mm

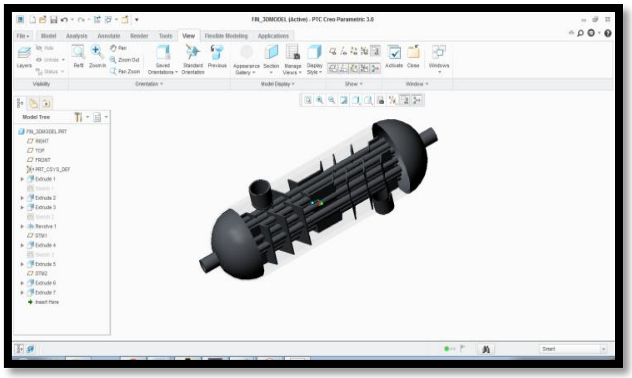


Case 2: fin angle 30^{0} and fin pitch 4.0mm



3D MODEL





INTRODUCTION TO FINITE ELEMENT METHOD

Finite Element Method (FEM) is also called as Finite Element Analysis (FEA). Finite Element Method is a basic analysis technique for resolving and substituting complicated problems by simpler ones, obtaining approximate solutions Finite element method being a flexible tool is used in various industries to solve several practical engineering problems. In finite element method it is feasible to generate the relative results.

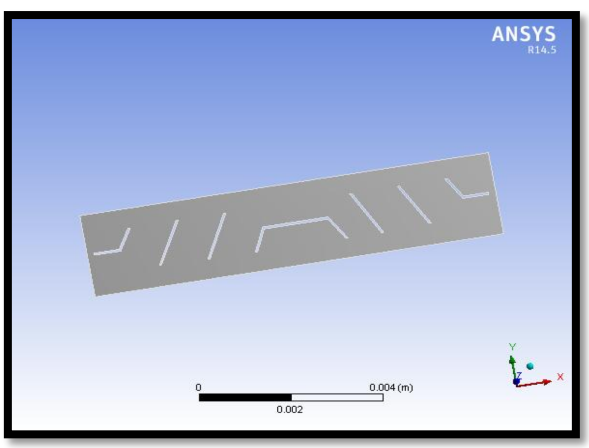
ANSYS Software:ANSYS is an Engineering Simulation Software (computer aided Engineering). Its tools cover Thermal, Static, Dynamic, and Fatigue finite element analysis along with other tools all designed to help with the development of the product. The company was founded in 1970 by Dr. John A. Swanson as Swanson Analysis Systems, Inc. SASI.

INTRODUCTION TO CFD

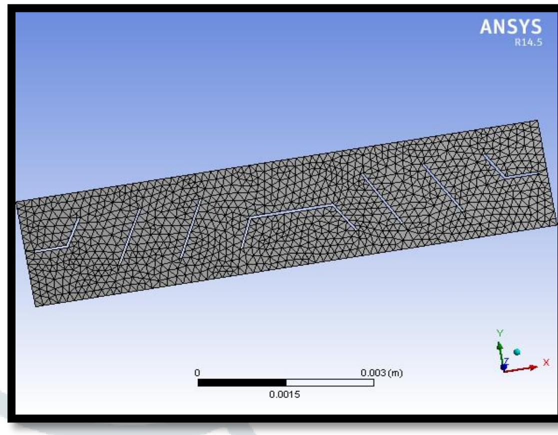
Computational fluid dynamics, usually abbreviated as CFD, is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved.

CFD ANALYSIS OF HEAT EXCHANGER FIN

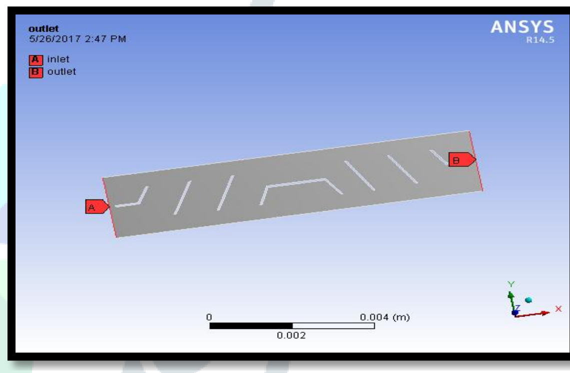
Case 1: fin angle 28^0 and fin pitch 1.0mm



MESHED MODEL



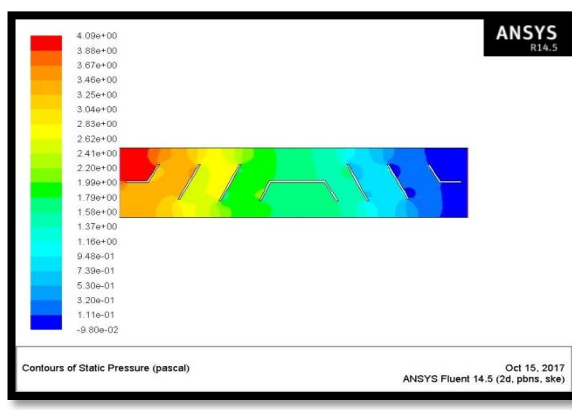
SPECIFYING BOUNDARIES FOR INLET AND OUTLET



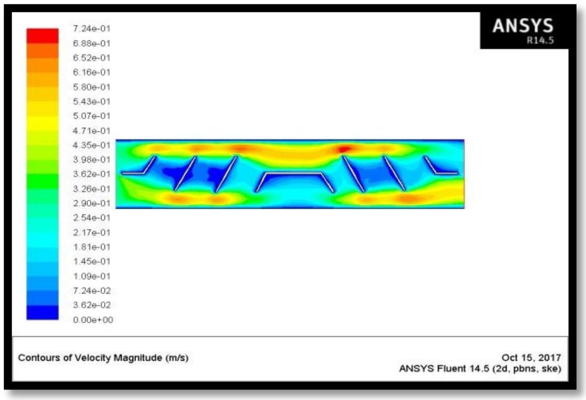
Boundary conditions → Inlet → Edit
velocity 15 m/sec–pressure -101325 pa
Solution → Solution Initialization→ Hybrid Initialization →done
Run calculations → No of iterations = 100 → calculate → calculation complete

Case 1: fin length 0.009m
Angle 28^0

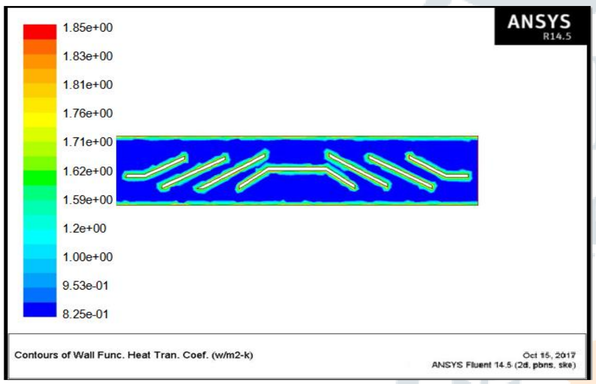
Pressure



Velocity



Heat transfer coefficient



Mass flow rate & Heat transfer rate

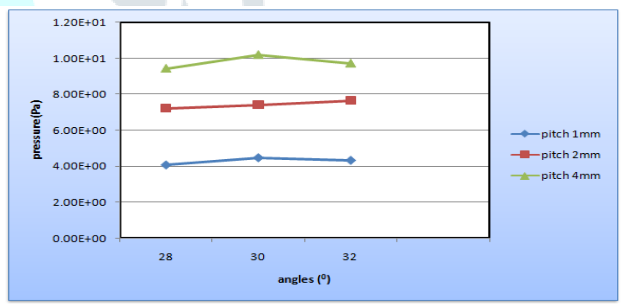
```
            Mass Flow Rate                  (kg/s)
-------------------------------- --------------------
                     inlet        0.00070496299
        interior-_trm_srf        -0.00099855475
                    outlet       -0.00070443825
           wall-_trm_srf                      0
        ---------------- --------------------
                       Net          5.2474206e-07

     Total Heat Transfer Rate                (w)
-------------------------------- --------------------
                     inlet            10.536011
                    outlet           -10.528211
           wall-_trm_srf                      0
        ---------------- --------------------
                       Net           0.0078001022
```

RESULT TABLE

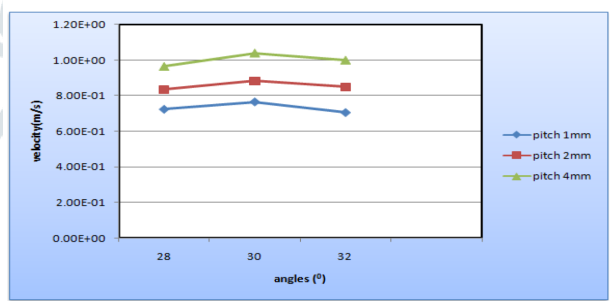
Fin pitch (mm)	Angle (0)	Pressure (Pa)	Velocity (m/s)	Heat trnsfer coefficient (w/m2-k)	Mass flow rate(kg/s)	Heat transfer rate (W)
1mm	28	4.09e+00	7.24e-01	1.85 e+00	5.2474e-07	0.0078001022
	30	4.48e+00	7.65e-01	1.91 e+00	8.734e-07	0.0131282
	32	4.34e+00	7.06e-01	1.20 e+00	7.8580e-08	0.0013198
2mm	28	7.23e+00	8.36e-01	1.21 e+00	4.1111e-06	0.061322
	30	7.42e+00	8.82e-01	1.219 e+00	3.904e-06	0.058273
	32	7.64e+00	8.48e-01	1.22 e+00	2.8242e-06	0.042238
4mm	28	9.43e+00	9.65e-01	1.29 e+00	9.3101e-06	0.13922
	30	1.02e+01	1.04e+00	2.03 e+00	5.459e-06	0.08139
	32	9.72e+00	1.00e+00	2.23e+00	6.2752e-06	0.09373

GRAPHS

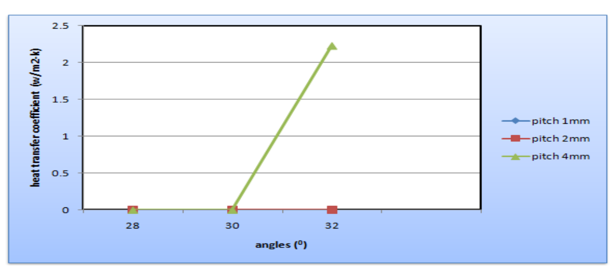
Pressure plot



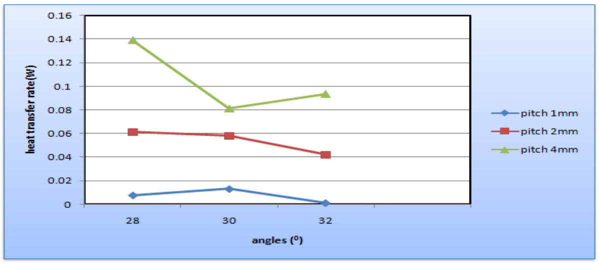
Velocity plot



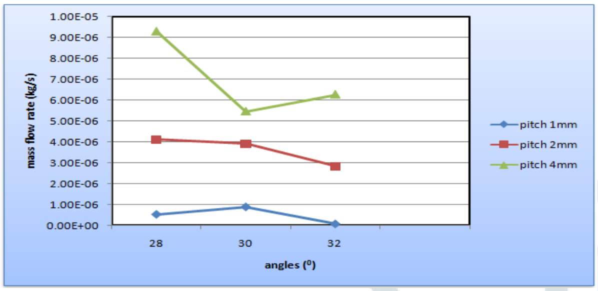
Heat transfer coefficient plot



Heat transfer rate plot



Mass flow rate plot



CONCLUSION

In this thesis the parametric analysis on thermo-hydraulic performance of a compact heat exchanger using computational fluid dynamics (CFD). The analysis has been carried out at different frontal air velocities by varying the geometrical parameters such as louver pitch and louver angle.

In this thesis the CFD analysis was performed the heat transfer rate, mass flow rate, pressure drop and velocity at 0.4887m/s inlet velocity.

By observing the CFD analysis results the heat transfer rate, mass flow rate values are more at 30^0-2^0 fin angle and fin pitch 4mm.

So it can be concluded the 30^0+2^0 fin angle and fin pitch 4mm is better performance for fin in heat exchanger.

REFERENCES

DESIGN AND FLUID FLOW ANALYSIS OF HEAT EXCHANGER WITH DIFFERENT LOUVER FINS

Indala. HariBabu [1],
M-Tech Student(Thermal engineering), Department Of Mechanical Engineering, , SITAM
Engineering College, pm palem Visakhapatnam, andrapradesh India[1]
CH. Venkata Anvesh [2]
Assistant Professor, Department Of Mechanical Engineering, , SITAM
Engineering College, pm palem Visakhapatnam, andrapradesh, India[2]

1. The Effect of Geometrical Parameters on Heat Transfer Characteristics of Compact Heat Exchanger with Louvered Fins.
2. Experimental and parametric studies of a louvered fin and flat tube compact heat exchanger using computational fluid dynamics.
3. Experimental and numerical investigation of a louvered fin and elliptical tube compact heat exchanger.
4. Advanced numerical simulation of compact heat exchangers. Application to automotive, refrigeration and air conditioning industries.
5. Comparative Study of Heat Transfer Enhancement in Rectangular And Interruped Louvered Fins (Newly Designed) in Internal Combustion Engine Using Cfd Tool
6. 3d-cfd simulation and neural network model for the j and f factors of the wavy fin-and-flat tube heat exchangers