KEMENTERIAN PENDIDIKAN TINGGI

JOJAPS

eISSN 2504-8457

POLITEKNIK MALAYSIA PORT DICKSON

Journal Online Jaringan Pengajian Seni Bina (JOJAPS)

Simulation of New Design Radiator Water Flow in CFD

Ts. Siti Rahaida Abdullah & Ts. Firdaus Ali

[a]Mechanical Engineering Department, Politeknik Ibrahim Sultan, KM 10, Jalan Kong Kong 81700 Pasir Gudang, Johor Darul Takzim, Malaysia

Abstract

Computational Fluid Dynamics (CFD) is used to simulate airflow through a model with two porous. This paper investigates the axial flow fan's flow characteristics, used for radiator cooling, using the CFD code FLUENT 6.0. The simulation was based on the formulation of the unsteady-state concentration to assess the ability of the CFD tool for the development of fluid flow and water flow through the radiator. The thermal performance of automotive radiators plays a vital role in vehicle cooling systems and all other related systems. This study is to determine the level of efficiency for thermal radiators in the form of squares. The radiator's surface cannot pass through the air velocity due to the housing is not filled. The principles of heat transfer and its search related to compact heat exchangers' design will form the basic knowledge that will be mastered by the principles of fluid mechanics and thermodynamics for this study. CFD modeling is a critical design tool to investigate significant problems related to radiator performance, such as mass flow rate and heat transfer, among others. The magnitude of the temperature at the inlet was set at 368K. Also, the surface pressure distribution was simulated in 2 Dimensional.

Key-word: - CFD, Fluent, Gambit, Radiator

1. Introduction

The heat dissipation capacity of an automated vehicle on converters' performance such as radiators that emit heat from the engine and condenser to provide comfort to the passenger compartment. The heat exchanger design consists of a fin layer with a louver embedded in a flat tube row. This configuration forms the basis of the radiator design. This is the geometric arrangement of the fins and the detailed tubes that determine the heat dissipation performance.

The determination of heat transfer behavior through a heat exchanger is mostly undocumented. Experimental observation is still under research but very expensive due to the small complex geometries in the heat exchanger design. With computational simulation and CFD modeling, this problem can be resolved. Experimental observation will still be required but to a lesser extent. With flow and heat transfer characteristics known, a new design can be developed and tested. Computational fluid dynamics (CFD) was used in this study to perform an optimization on the louver geometry using a commercially available CFD code in parallel with optimization software.

In this paper, Computational Fluid Dynamics (CFD) based numerical approaches were used. Radiator design is a secret factor of industrial trade in actual production. This study focuses on the design of radiators in experiments of real output. With the increasingly sophisticated design of vehicles, there is a greater demand for compact heat exchanger units for engine cooling and passenger comfort. This research will provide the necessary data and knowledge leading towards the development of new heat exchanger designs that will take place for the national automotive industry and heat exchangers worldwide.

2. Methodology

2.1 Introduction

A computational effort was undertaken to analyze the details of fluid flow in the various components of a radiator panel of the CEV Service Module. The effort focused on determining the envelope of environmental conditions and flow conditions at which the flow in the radiator would approach "stall" or near-freezing situations. The FLUENT code was used to perform conjugate heat-transfer computations for a single radiator panel with a face-sheet and upstream and downstream headers. The single panel computations were used extensively to provide feedback to designers performing thermal analysis with the Thermal Desktop software package. The CFD analysis offered several insights into the correlation between "stall" areas in the radiator and coolant mass flow rate.

This research aimed to simulate the built experimental testing setup to test the performance characteristic of various automotive radiator designs under simultaneous actual heat dissipation loading and different climatic conditions. Measure the flow characteristics of automotive compact heat exchanger units under real conditions using experimental techniques. Also, stimulate the flow characteristics of automotive compact heat exchanger units under actual conditions using computer simulation techniques. The steps taken to achieve this objective are depicted in the accompanying areas of this section.

2.2 Model geometry development software

The principal objective of the CFD simulation was to get the best simulation that was directly similar to airflow in the real physical phenomenon so that the results obtained from the models could be approved. A hidden objective was to expand the learning of the methodology.

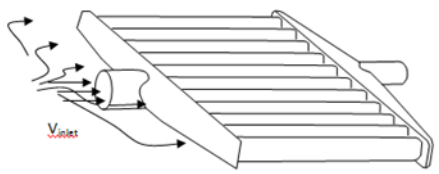

Figure 1 Rectangular radiator

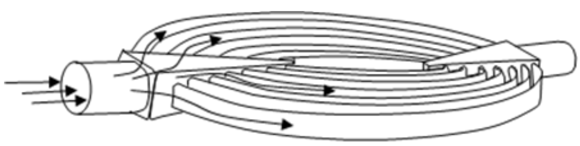

Figure 2 Circular radiator

2.2.1 Discarded geometry and mesh

The 3D drawing of the model was as shown in Figures 3 and 5. It is made to fit the computer's capability as complex, and large dimensions would require a high processing capabilities computer and longer meshing and simulation time. A 3D CAD modeling of a rectangular radiator and circular radiator was build using AutoDesk AutoCAD, as shown in Figure 4 and Figure 6. This is to determine the overall size for the radiator with the same water volume.

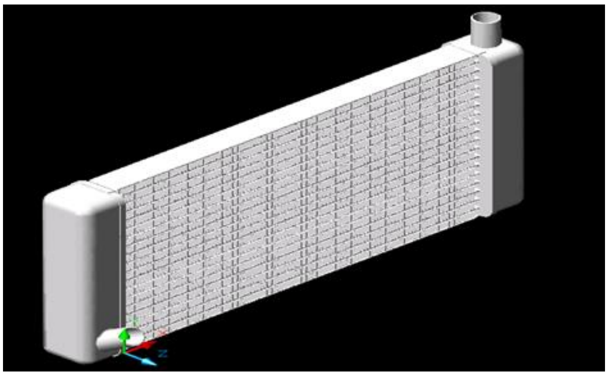

Figure 3 3D Drawing Mesh Rectangular

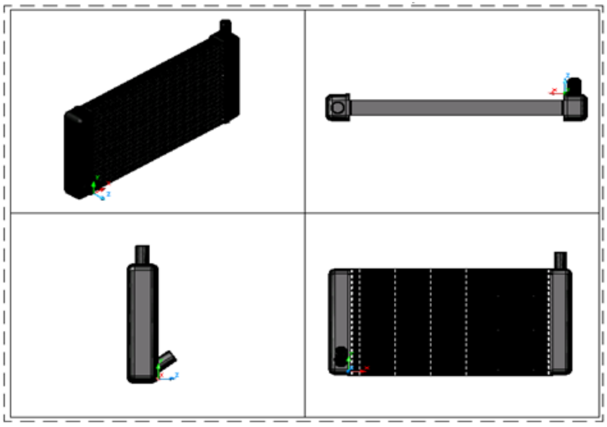

Figure 4 CAD Model Drawing Mesh Rectangular

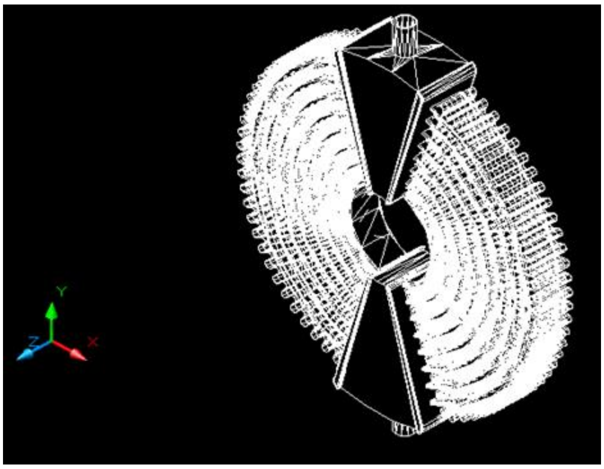

Figure 5 3D Drawing Mesh Circular

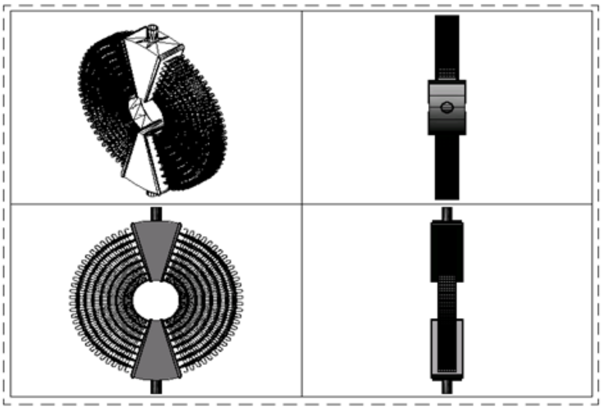

Figure 6 CAD Model Drawing Mesh Circular

The CFD process begins with 3D geometry created using a CAD package in the present work. The geometry then forms the framework around which a mesh would be constructed, and the relevant conditions would be set up. Initially, the models consisted of 2D after reading the case (mesh file 40096 cells). The model of several parts of energy, material database, water liquid, and outflow. For the circular radiator inlet, 0.1 m/s, temperature 368K, wall thickness is 0.0005 (assume hot water in the tubes being cooled by the wall due to cold air at 20°C blowing outside). The hydraulic diameter - dh - is used to calculate the dimensionless Reynolds Number to determine if a flow is turbulent or laminar. A flow is :

i. laminar if Re < 2300
ii. transient for 2300 < Re < 4000
iii. turbulent if Re > 4000

Note that the flow velocity in the Reynolds calculation is based on the actual cross-section area of the duct or pipe. The hydraulic diameter is also used to calculate the pressure loss in a duct or pipe. The hydraulic diameter is not the same as the geometrical diameter in a non-circular duct or pipe and can be calculated with the generic equation:

$$dh = 4\,A / p \qquad \text{(Eq. 2.1)}$$

where,

dh = hydraulic diameter (m, ft)
A = area section of the duct (m2, ft2)
p = wetted perimeter of the duct (m, ft)

The model was then rebuild using CAD software called GAMBIT. GAMBIT was selected as the geometry builder because of the ability to do the mesh and boundary condition setup, which were used in the model's simulation in FLUENT. The model was simulated in 2 phases. The first phase was a simulation in 2 Dimensional. Therefore, a model was built for each dimensional in GAMBIT. Figure 7 shows the wireframe schematic of the model in 2 Dimensional.

Those meshed models were then exported as a mesh file, which then will be read by FLUENT 6 to begin the simulation. The meshed models could be exported in various types of file formats such as ACIS, CAD, etc., depends on the simulation software used for the simulation process. For example, if using ANSYS FLUENT, ACIS type of file format would be the best to represent and export the meshed model.

2.2.2 Boundary condition

Boundary conditions for the model built in GAMBIT were as below:

Table 1 Boundary conditions

Dimensional	Edge/Face/Volume	Boundary Condition	
2 Dimensional	Inlet (Face)	Velocity Inlet	0.1 (m/s) Temp = 368 K
	Wall	Thickness	0.0005m Temp = 293K
	Outlet (Face)	Pressure Outlet	Temp = 293K

2.2.3 Solution of the model

Initializing the solution of the model was the final step towards solving the two-phase flow problem. The following sections describe the various parameters that had to be specified to initialize a solution and how each parameter was chosen.

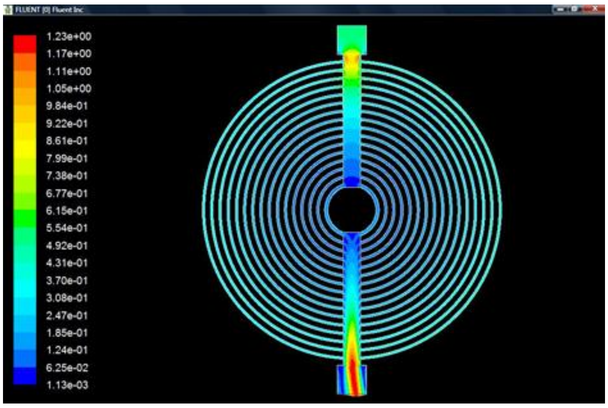

Figure 7 Velocity vectors for 30 tubes circular radiator.

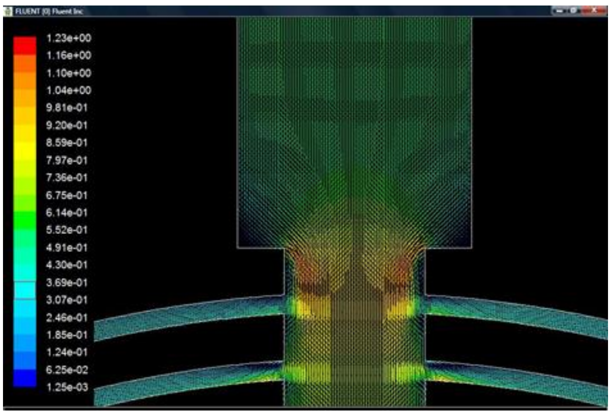

Figure 8 Velocity of water flow at the header.

3. Result and discussion

3.1 Introduction

This study shows that a complex two-dimensional CFD simulation of the flow through the circular radiator's front-end is possible under a variety of conditions. The accompanying validations have shown that the result is accurate for realistic operating conditions. This opens the door to the possibility to optimize the cooling system design early in development. The next step is to incorporate heat transfer effects in the radiator and a tighter coupling of the simulation with the heat transfer model.

3.2 Comparison of 2D circular radiators and rectangular radiators

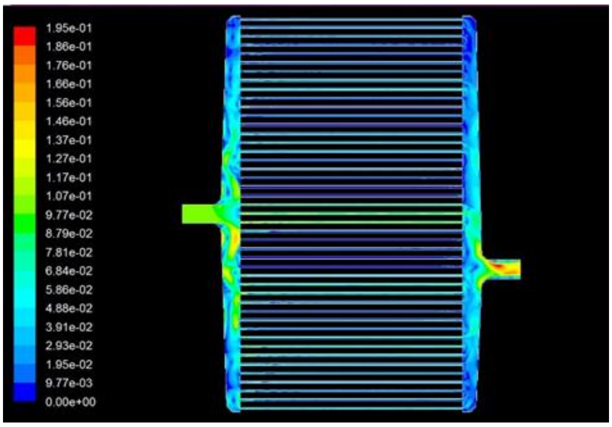

Figure 9 Circular CFD analysis for water flow at 1ms-2

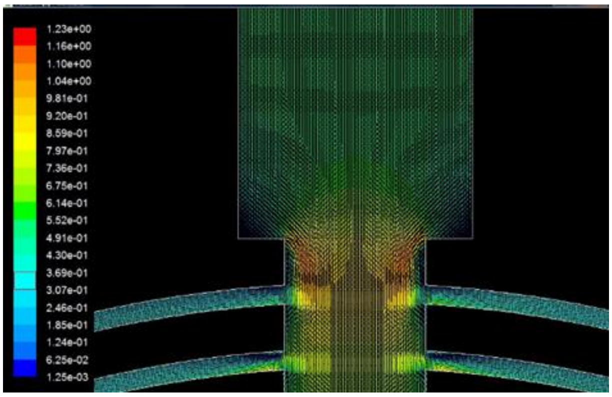

Figure 10 Radiator CFD analysis for water flow at 1ms-2.

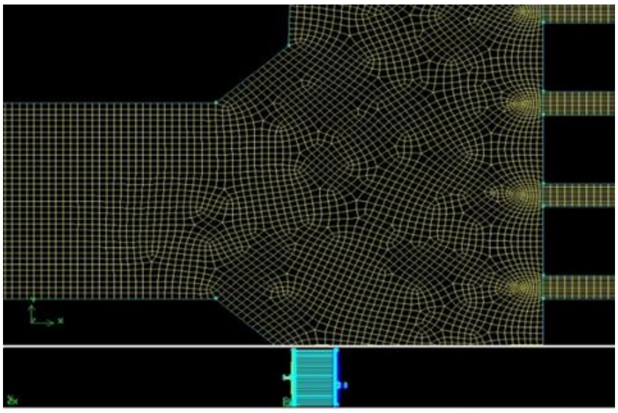

Figure 11 Zoom meshed face of a rectangular radiator (inlet to a tube)

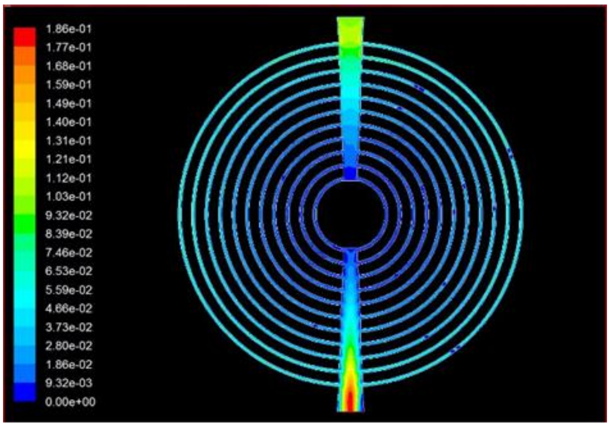

Figure 12 Circular CFD analysis for water flow at 1ms-2.

A computer simulation program has been developed for predicting the thermal performance of a vehicle cooling system. It has been shown that there is good agreement between experimental and expected results, thus providing confidence to use the model as an analytical tool for a new vehicle. The radiator result shows excellent agreement with the experimental data.

4. Conclusion

CFD enabled optimization of circular radiator cover was carried out. Starting from a validation solution of the baseline design, optimization of the radiator was carried out using iterations. The final optimized configuration analysis shows the elimination of recirculation zones and increased flow through the radiator. This allows an efficient application of CFD to a wide variety of developmental problems. While it is undoubtedly possible to delve into more rigorous and detailed analyses, it has been our experience that the types of projects reviewed in this paper lend themselves well to more superficial CFD analyses, thereby allowing either design direction or understanding of the flow field to be realized. This, in turn, will enable CFD to be applied as efficiently as feasible to automotive development.

The expected and possible values about the new proposed heat exchanger are low-velocity zones, high-temperature regions (low heat transfer region) identified in corners. This study presented the spatially resolved heat transfer coefficients for rectangular geometries. The heat transfer results from the current study gave a representative experimental database of several geometries. For optimum efficiency, eliminate corners and develop a radiator of circular shape and design that is compact.

References

Abhishek Gakare et.al (2019) "Simulation Software Based Analysis of Automotive Radiators: A Review" Journal of Automation and Automobile Engineering, Vol 4(1), pp13-18

C. Uma Maheswari et.al (2016) "Optimization of Design Parameter of Radiator" Energy, Economy and Environment 2016.

David J. Kukulka, Kevin G. Fuller (2010). "Development of an Enhanced Heat Transfer Surface.". 20th European Symposium on Computer Aided Process Engineering.

N. Nagarani, K. Mayilsamy (2010). "Experimental Heat Transfer Analysis on Annular Circular and Elliptical Fins.". International Journal of Engineering Science and Technology, Vol. 2(7), pp. 2839-2845.

P. Gunnasegaran, N.H Shuaib, M.F Abdul Jalal, E. Sandhita (2012). "Numerical Study of Fluid Dynamic and Heat Transfer in a Compact Heat Exchanger Using Nanofluids.". International Scholarly Research Network, Vol. 2012.

Prof. Chavan Dattatraya K, Prof. Dr. G.S Tasgaonkar (2012). "Analysis-Design and Modification of A Conventional Fan Assisted Heat Exchanger (Radiator) For Achieving Optimum Thermal Characteristics.". International Journal of Engineering Science and Technology (IJEST).

Prof. D.K. Chavan, Prof. Dr. G.S. Tasgaonkar (2011). "Thermal Optimization of Fan Assisted Heat Exchanger (Radiator) by Design Improvements." International Journal of Modern Engineering Research (IJMER), Vol. 1, Issue 1, pp. 225-228.

S.SJoshi,V.M.Kriplani (2011). "Experimental Study of Heat Transfer in Concentric Tube Heat Exchanger with Inner Twisted Tape and Annular Insert". International Journal of Advanced Engineering Sciences and Technology, Vol.10, pp. 334-340.