

# CFD Modelling of Air Flow Distribution from a Fan

Mohd Yusoff Sulaiman, Shamsul Bahari Azraai, Wan Mokhtar Wan Abdullah  
Faculty of Mechanical Engineering  
Universiti Teknikal Malaysia Melaka, Malaysia  
shamsulbahari@utem.edu.my

**Abstract**— This paper presents a Computational Fluid Dynamics (CFD) modeling simulation of air flow distribution from an automotive radiator fan. An introduction to the nature of the physical fan air flow problem and its significance was elaborated in order to understand the complications involved in the research and thereafter arrive at the objectives. Knowing the physical situation is crucial in the application of CFD to numerically model and thereby analyse the simulation. The tasks undertaken to model the geometries of the fan and its surrounding is the first important step. This is followed by meshing and defining the boundary conditions before numerically solving the variables that represent flow fields of the simulation. The numerical predictions of the variables in the form of velocity vectors and contour plots detailing the flow characteristics are then analysed, compared and verified according to known physical situation and existing experimental data. The result shows that the average velocity outlet air velocity is 10 m/s. The error of average outlet air velocity is 12.6 % due to dissimilarity in the tip shape of the blade. This study has shown that the CFD simulation can be useful tool in optimizing the design of the fan blade

**Keywords** – CFD Modeling, Automotive Radiator Fan, and Air Flow Distribution,

## I. INTRODUCTION

An automotive radiator fan and fan shroud, Figures 1 and 2, form a crucial role in the complicated automotive engine cooling system, Figure 3. The automobile burns fossil fuel in an engine that produces torque and movement. Waste heat produced is absorbed by the coolant in the radiator and transferred to the surrounding air. A radiator, [1] shown in Figure 4, is therefore a type of heat exchanger designed to transfer thermal energy from the coolant to the surrounding air via a mechanism known as natural or forced convection. The

latter case concerns the use of a radiator fan to pull the air through the radiator core which is partly flattened tubes but mainly flat plate louvered fins forming the heat transfer surfaces.

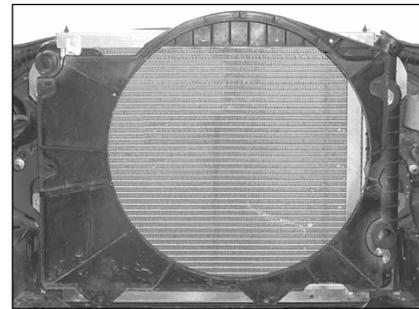


Fig. 2 A radiator and fan shroud assembly.

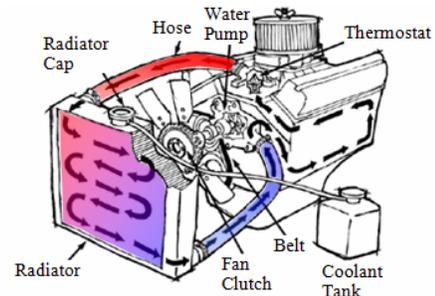


Fig. 3 A schematic of a cooling system.

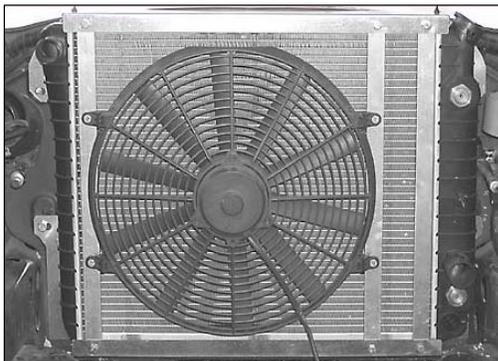


Fig. 1 A radiator and fan assembly.



Fig. 4 A automotive radiator

TABLE 1  
 FACTORS AFFECTING THE PERFORMANCE OF AUTOMOTIVE ENGINE COOLING SYSTEM.

Factors	Outcomes	Remedy
Air flow too slow	Coolant temperature overheats	Ensure air passages not clogged, change to higher capacity fan
Air flow leakages	Poor cooling ability	Fit proper shroud
Coolant flow too slow	Cannot carry enough heat away in timely manner	Change to a higher capacity coolant pump
Coolant flow too fast	Does not have time to be cooled in the radiator	Add heat dissipating additive or use flow restrictor
Hot weather spells, climbing hill, towing	Engine can overheats	Change to 2-rows or 3-rows radiators with improved air flow fans
Corroded, damaged, missing, loose fins, blocked air passages	Coolant temperature overheats in the engine section	Service or replace radiator
Thermostat not functioning	Coolant flow is blocked	Service or replace thermostat, flush coolant, use fresh clean coolant
Higher output engines replacement	Poor thermal performance for system cooling	Use integrated cooling system

There many different radiator designs and based on different materials: 1 to 4 tube rows, fins that are continues, louvered, straight, serpentine, tubes that are rounded, dimpled, flattened, cross flow, down flow, 1-pass core, 2-pass core, copper, brass, aluminium, etc. Figure 5 shows three different designs of the radiator core. The larger the size of a radiator design, the bigger the air flow rate. Figure 6 shows a twin fan and housing assembly. Integrated Cooling systems are available commercially, Figure 7, that combines the best performance of different components. However no one design can guarantee no engine overheating. For the engine cooling system, the contributing factors to the unwanted outcomes are listed in Table 1.



Fig. 6 Twin Electric Fan and Housing [3].

Automotive HVAC system utilized for heating and cooling of passenger compartments is directly connected to the engine cooling system. The components of this system are shown in Figure 8. One of them is the condenser, located in front of the radiator. Numerical modeling of this configuration is still under study, Yang [6]. Both the radiator and condenser share the function of the fan for assisted air flow. This involves the control of three major components which are water flow in the radiator, coolant flow in the condenser and the fan air flow. Several new systems have been developed for example Birch [7] wrote about THEMIS, a system capable of managing and optimizing engine temperature according to the driving conditions. It features electrical and electronic devices to control the water pump, fans speed, and coolant valve, replacing the conventional thermostat. Engine warm-up time is claimed to reduce by up to 50% with enhancement in cabin comfort.



Fig. 7 Integrated Cooling System [4].



(a) (b) (c)  
 Fig. 5 Radiator Core Design [2]: (a) Wide Tube, (b) Two-row narrowly spaced, (c) Two-row loosely spaced.

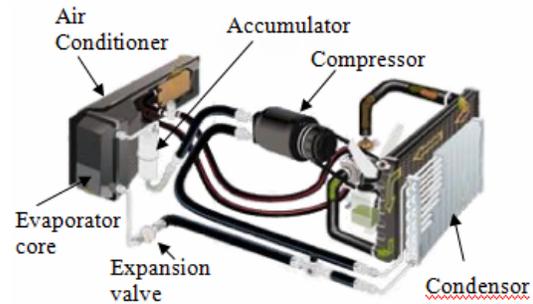


Fig. 8 An automotive HVAC system [5].

## II. NUMERICAL METHOD

With all the practical knowledge and awareness of the vehicle engine cooling system elaborated in the previous section, a logical conclusion was to undertake a task which can address the problems related to the cooling system. Central to the issue is the understanding of the flow distribution passing through the system. In this paper, CFD was utilized to model the flow passing through a radiator fan. Without the understanding of air flow characteristics passing through the fan, the problems relating to engine cooling systems can never be fully resolved.

The first step is to identify a typical automotive radiator fan that can be reproduced as a 3-dimensional CAD Solid Works engineering drawing package. There are many different sizes and designs available but only one simple design is chosen for the purpose of this paper, Figure 9. The drawings are then imported into the CFD software, remodeled into different sections and refined to generate a finite volume meshing. This is a crucial step where details to the geometrical shape need to be defined precisely. The flow domain is also created and the final meshing of all components need to be accurate. Any errors in the drawings and flow area will need to be corrected. The next stage is to import the files into the CFD Code preprocessor ready for solving the flow equations. Here the flow fields boundary condition are set. These include inlet air flow velocity, outlet pressure, fluid properties and flow domain characterization such as moving fan blades and stationary solid walls. The next step is to set the simulation process as a 3-dimensional, steady and laminar problem. The simulation is preceded with the CFD code processing the data, applying the basic theory of fluid mechanics by balancing the mass continuity and momentum equations in numerical form and thereafter producing numerical predictions of the flow variables. This iteration process needs to be monitored for convergence and repeated if the numerical error conditions are not satisfied. The final step is to analyse the output data and present them in the form of velocity vector distribution and contour plots.



(a)



(b)

Fig. 9 A four bladed automotive radiator fan, AF0011, HeatTransfer Research Unit, Faculty of Mechanical Engineering, UTeM, (a) inlet view, (b) outlet view.

### A. Numerical CFD Model

The fan geometrical parameters have been chosen as shown in Table II. Figure 11 shows the CFD flowchart from construction of geometry to post processing. The CAD fan model, Figures 9 and 12 were imported into preprocessing software for regional identification, surface meshing and finally grid generation, Figures 13 to 16. The physical model was assumed to be a fan suspended in a cylindrical ducting as shown in Figure 17. The flow domain representing the cylindrical duct was created and meshed accordingly, Figure 17 with  $G = 400$  mm,  $H = I = 200$  mm. The numerical model is then imported into the main CFD programme where the flow domain is transformed into a CFD finite volume model for solving the flow physics. The problem setup process is completed by defining the boundary conditions, solver controls and convergence monitors. The boundary conditions include using steady laminar incompressible flow, dry air at standard atmospheric pressure, fixed wall, and moving fan at 1500 rpm, zero pressure at outlet and inlet velocity set for 5 m/s. The segregated solver with an implicit formulation was used. The residual values of all variables solved were monitored during the iteration process with mass balance set to less than  $1.0E-6$ .

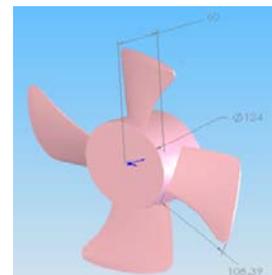


Fig. 10 Isometric view of CAD fan.

TABLE II.

FAN GEOMETRY

Section	Dimension (mm)
1. Fan Blade	
Thickness	2
Neck width A	64.5
Tip width B	117
Length C	109
Length D	98
Width E	29
Width F	30
2. Fan Hub	
Diameter	124
Thickness	60

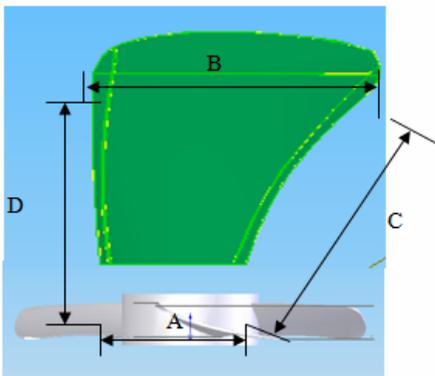


Fig. 13 Characteristics geometry of fan.

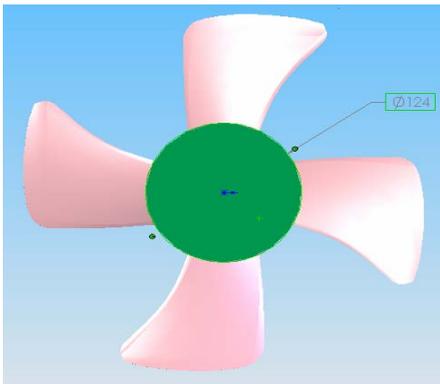


Fig. 12 Front view of CAD fan.

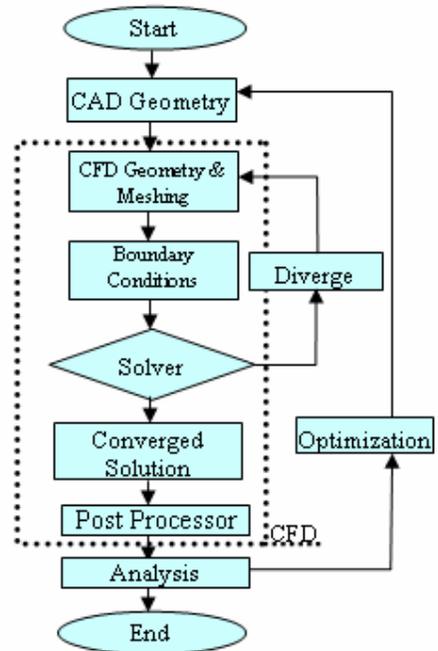


Fig. 11. Methodology flowchart of CFD modelling.

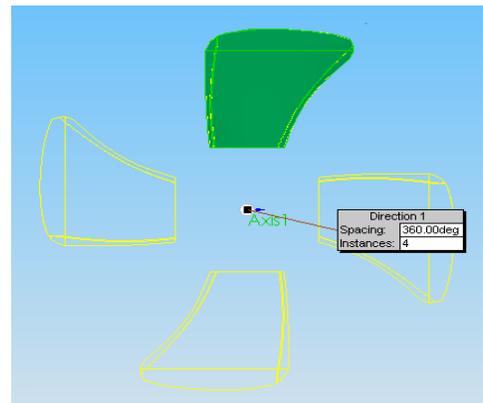


Fig. 14 Numerical CFD fan geometry.

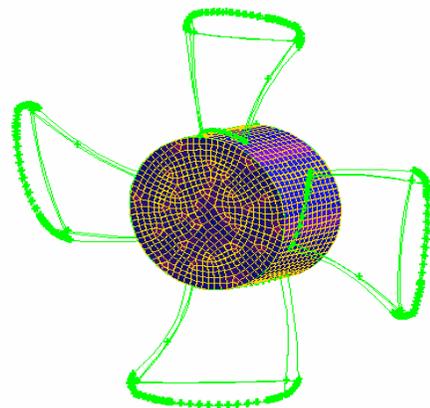


Fig. 15 Meshing CFD fan geometry.

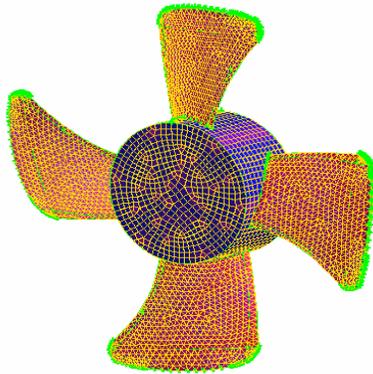


Fig. 16 Complete mesh of CFD fan geometry

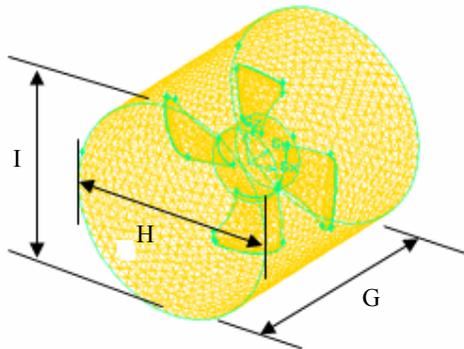


Fig. 17 Complete mesh of CFD fan and flow domain geometry.

### III. NUMERICAL RESULTS AND DISCUSSION

On post processing the numerical CFD results, the following were the observations presented as velocity vector distributions. Refer Figure 18 for the cross-section between fan blades and Figure 19 for the cross-section cutting the fan blades. These two figures show the cross-section of the flow from upstream to downstream section. The cross-section views perpendicular to the flow are shown in Figure 20 and Figure 21.

#### A. Presence of two flow regions

High flow region was formed around the outer diameter of flow domain. Low reverse flow region was formed in the centre behind the fan hub. These observations are similar to that reported by Chacko [8] although the locations are different due to dissimilar geometries.

#### B. Presence of re-circulation regions

Between the high and low reverse flow regions, there exist strong circulation vortices. Strong circulations regions were also observed behind the fan blades. Since the fan blades were rotating and the fact that the flow was downstream, there would be a number of downstream moving, circulating vortices. This prediction is similar to the particle path lines reported by Park [9].

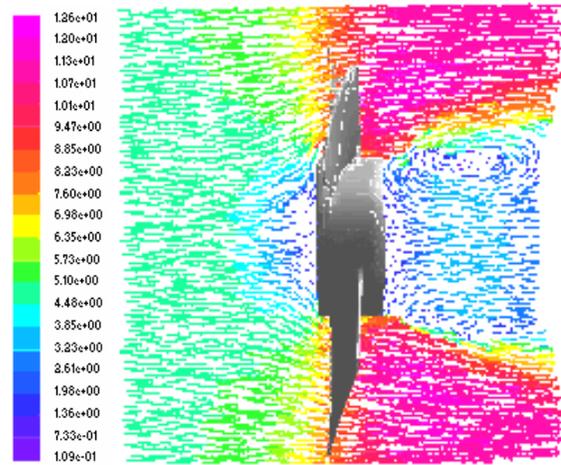


Fig. 18 Velocity vectors between blades.

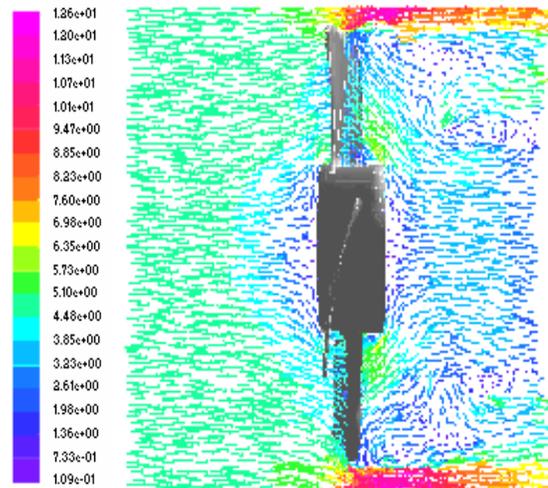


Fig. 19 Velocity vectors across blades.

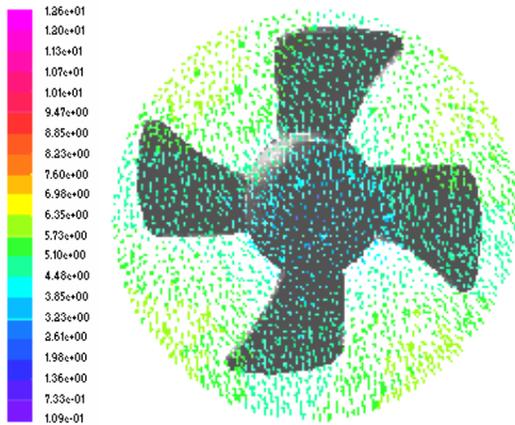


Fig. 20 Velocity vectors 60mm upstream of fan.

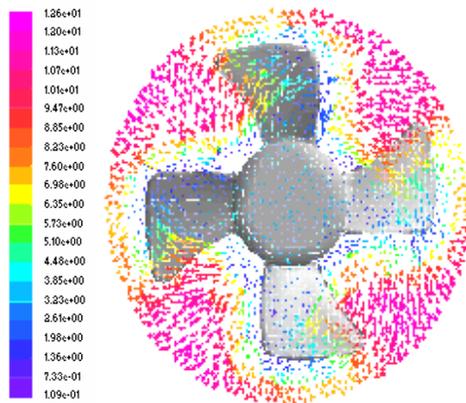


Fig. 21 Velocity vectors 60 mm downstream

Another way to present the numerical results is to plot all the magnitude of the velocities in the flow domain against the flow length, Figure 22. The plots showed that the inlet velocity of 5 m/s began to increase or decrease slowly at first but rapidly as the flow cross the fan and thereafter was mainly distributed either at the high flow region or at the low reverse flow region. While the profile in velocity increase for maximum values are smooth, there appeared to be a break the profile of velocity decrease for minimum values that occurred at the fan configuration. This phenomena could be attributed to re-circulation and vortex formation behind the fan blades. Further detailed analysis would be required to verify this phenomena. The horizontal line profile at zero velocity can be attributed to no slip condition near the duct wall.

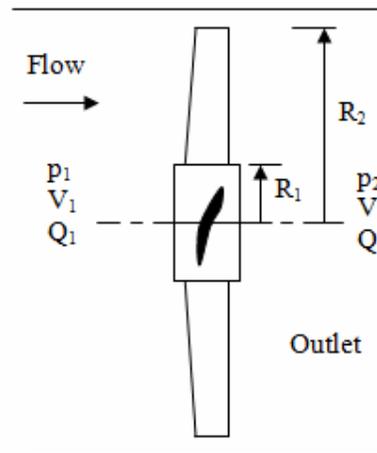


Fig. 23 Calculation parameters.

C. Calculations

The air flow on the axial flow fan blade can be performed using the velocity triangle equations for the device as shown in Figure 23 and Figure 24. The flow medium is air, density,  $\rho = 1.225 \text{ kg/m}^3$ , inlet velocity,  $v_1 = 5\text{m/s}$  and the fan speed,  $N = 1500 \text{ rpm}$  or  $\omega = 157.0797 \text{ rad/sec}$ . Air inlet area,  $A = \pi j^2 = \pi \times (0.18 \text{ m})^2 = 0.10179 \text{ m}^2$ .

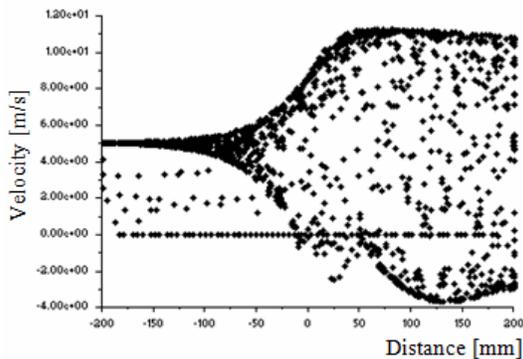


Fig. 22. Plots of velocities against flow distance.

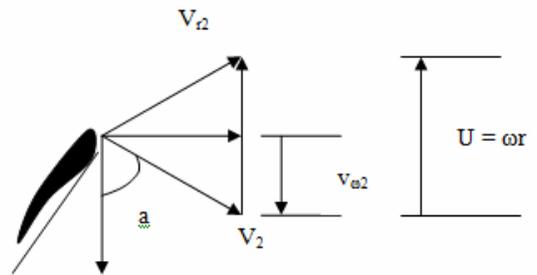


Fig. 24 Velocity triangle for the fan blade.

For inlet condition,  $Q_1 = V_1 \times A$   
 $= 5 \text{ m/s} \times 0.10179 \text{ m}^2$   
 $= 0.509 \text{ m}^3/\text{s}$

Tip velocity,  $u_t = \omega r_1$   
 $= 157.0797 \text{ rad/s} \times (0.109 \text{ m})$   
 $= 17.12 \text{ m/s}$

Applying dimensional analysis theory:  $Q_1^2 = g H_{\text{water}} D^4$   
 where  $D = D_{\text{blade}} + D_{\text{hub}}$   
 $= 0.342 + 0.124$   
 $= 0.466 \text{ m}$

Then,  $H_{\text{water}} = Q_1^2 / g D^4$   
 $= 0.509^2 / 9.81 \times 0.466^4$   
 $= 0.56 \text{ m}$

Due to similar height for blade and hub, then the outlet angle,  $\beta_{2t}$  can be calculated by applying Euler equation :

$$E = (u_t/g)(u_t - v_f \cot \beta_{2t})$$

where  $V_1 = v_{f1} = v_{f2} = 5 \text{ m/s}$

Then,  $E = H_{\text{water}} \times \rho_{\text{water}} / \rho_{\text{air}}$   
 $= 0.56 \text{ m} \times (1000 \text{ kg/m}^3 / 1.225 \text{ kg/m}^3)$   
 $= 457.14 \text{ m}$

We have,  $E = (u_t/g)(u_t - v_f \cot \beta_{2t})$   
 $457.14 \text{ m} = (17.12 \text{ m/s} / 9.81 \text{ m/s}^2) (17.12 \text{ m/s} - 5 \text{ m/s} \cot \beta_{2t})$   
 $\beta_{2t} = 88.02^\circ$

Then,  $v_{w2} = u_t - v_f \cot \beta_{2t}$   
 $= 17.12 \text{ m/s} - 5 \text{ m/s} \cot 88.02^\circ$   
 $= 13.52 \text{ m/s}$

Therefore the outlet velocity at the blade is,

$$V_2 = \sqrt{(13.52^2 + 5^2)}$$

$$= 14.4 \text{ m/s}$$

The outlet velocity is a point located at the tip section of the blade. From the simulation result, the maximum velocity at the tip of the blade is 12.5921 m/s.

Therefore the error is,  $= \frac{14.4 \text{ m/s} - 12.5921 \text{ m/s}}{14.4 \text{ m/s}} \times 100\%$   
 $= 12.6 \%$

The error is rather large but this can be attributed to the dissimilarity in the tip shape of the blade where the numerical model have rounded tip but the theoretical shape is a 90 degrees edge.

#### IV CONCLUSION

The CFD numerical modeling technique shown here in this study proves to be very useful in initiating further and more comprehensive numerical study of the engine cooling system which is currently in progress. Identification of high and low air flow regions with re-circulations and vortices

hugely improve our understanding of flow in the complex system involved. The design requirement from automotive manufacturer for slim line radiator fan has encouraged fan manufacturers to design this fan accordingly but with bigger diameter fan hub. This study has shown how the flow is interrupted by the hub obstruction thereby resulting in unwanted reverse flow regions. A simple solution yet to be verified is to design a slightly aerodynamic shape of the fan hub. This study has shown how CFD modeling can be a useful tool in optimizing the design of a component critical to the overall function of a complicated engine cooling system and cannot be substituted by the uncertainty outcomes of experimental methods. The traditional experimental approach is costly, requiring precise measuring equipments and time consuming. The poor outcomes of CFD analysis can easily be rectified to arrive at good objectives but with no significant extra cost.

#### ACKNOWLEDGMENT

The authors would like to thank the top managements of the Universiti Teknikal Malaysia Melaka for approving the funds required to undertake the research. The encouragements and supports have given the research team new ideas and opportunities to work upon and be top researchers of the current industrial design and development revolutionist at international level. Special thanks to the Experimental Heat Transfer Research Group and the Thermal-Fluids Technicians of the Mechanical Engineering Faculty for providing access and information to the heat exchanger units.

#### REFERENCES

- [1] Wikipedia, the free encyclopedia, "Radiator", Wikipedia® Wikimedia Foundation, Inc., 2006.
- [2] JTR, "Stealt Conversion 1979-1983 Datsun 280ZX Aluminium Radiator Kit", "V8 Datsun ZX, V8 Radiator, Datsun ZX V8 Conversion Radiator Kit", <http://www.JagsThatRun.com/>, 2005.
- [3] Ofria, C., "A Short Course on Cooling Systems", SmartTrac Computer Systems, Inc. 2006
- [4] EMP Stewart Components, "Integrated Cooling System", Stewart Components, Inc., 2006
- [5] Auto Heating & Air Conditioning Compressor, Condenser, Radiator Gemini Automotive Care.htm, "Automotive services, Heating & Air Conditioning", <http://www.goodyear tires.com/>, 2005.
- [6] Yang, Z., et. al., "CFRM Concept at Vehicle Idle Conditions", SAE Technical Paper Series 2003-01-0613, 2003
- [7] Birch, S., "Tech Briefs: Valeo and Ricardo ready the i-MoGen", Automotive Engineering Online, <http://www.sae.org/automag/techbriefs/05-2002/page2.htm>, 2002.
- [8] Chacko, S., et. al., "Numerical Simulation For Improving Radiator Efficiency", Engineering Automation Group, Tata Technologies Limited, Engineering Research Center, Tata Motors Limited, India, 2003.
- [9] Park, J., "A Sound Method for Fan Modeling", Fluent News, US, Summer 2005.

