Journal of Applied Sciences

ISSN 1812-5654

science alert

ANSInet
an open access publisher
http://ansinet.com

Journal of Applied Sciences 13 (11): 2027-2032, 2013
ISSN 1812-5654 / DOI: 10.3923/jas.2013.2027.2032

Correlation of Engine Cooling System Parameters and Segmented Heat Exchangers' Analysis

S.C. Pang, H.H. Masjuki, M.A. Kalam and M.A. Hazrat
Department of Mechanical Engineering, Faculty of Engineering, University of Malaya, 50630, Kuala Lumpur, Malaysia

Abstract: Current research performs mathematics correlations between engine speed, coolant flow, vehicle speed and driving gear. A step-by-step procedure is described to obtain the engine cooling system parameters mathematically (include a CFD model). After obtaining the parameters, the thermal equilibrium of engine cooling system is studied thoroughly. The study of thermal equilibrium provides some insights on how to reduce engine cooling load and when the interference of cooling fan is required. A segmented spread sheet model is developed in order to explain the phenomenon which air flow driven by uniform ram air could dissipate higher amount of heat flow than air flow driven by cooling fan. The segmentation analysis concluded that minimum mC_p fluid is switched to coolant when the air flow is concentrated at small portion of area.

Key words: Finite element analysis, cooling load, segmentation, gear, heat exchanger, heat transfer, underhood air flow

INTRODUCTION

In this research, the thermal equilibrium of engine coolant circuit is formulated in an equation. Engine heat and radiator heat will determine the net heat input for engine coolant circuit. The net heat input will either be utilized to increase coolant internal energy (coolant temperature increases) or to break the atoms' bonding (coolant boils). Later, this research describes a procedure to obtain the parameters of engine cooling system in stages. Firstly, we obtain a set of industry data constituted of engine speed, coolant flow and engine heat. Secondly, we correlate mathematically the industry data with vehicle speed and driving gear. With this correlation, it could be observed how the combination of vehicle speed and gear could vary the engine speed, engine heat and coolant flow. Thirdly, a Computation Fluid Dynamic Model (CFD) is established to determine the air flows' throughput at radiator and radiator heat. Industry provided hood's geometry is imported to CFD model and air flow pattern flowing through these complex geometries is simulated. From dual stream heat exchanger report in CFD model, we obtain radiator heat and perform further analysis.

In literatures, researchers in various fields tend to perform discretization in their computational. With discretization, the problem will be computed/calculated for every single nodes, segment and element. The output parameters which are obtained from the discretization could provide a contour picture of temperature distribution, stress distribution, flow distribution and etc. Researchers diced the area of a cavity in order to observe the temperature distribution without/with the movement Nano particles (Giraldo *et al*., 2011). Another popular term which researchers are using is finite element analysis. With element analysis, researchers avoid over-generalized computation, replaces single averaged equation with N-computational equations. Civil engineers also distributed the reservoir bed into elements, in order to study the geometry effect of reservoir bed towards the hydrodynamic pressure (Maity, 2005). In medicine field, researchers segmented the tapped shape human limb to investigate the heat flow in dermal region (Agrawal *et al*., 2010). Segmentation into element are performed in many other applications such as to study heat conduction in plate (Kaneko *et al*., 2008), to study stress distribution in brick (Kaveh and Tolou Kian, 2012) and to study the stress distribution among the gears in transmission system (Li *et al*., 2002)

The air flow distribution on the frontal face of heat exchangers could affect their thermal efficiency. Experiment studies and numerical studies were conducted to study the maldistribution of air flow towards air-cooled heat exchanger (Habib *et al*., 2009), industrial heater (Hoffmann-Vocke *et al*., 2011), evaporator (Kaern *et al*., 2011) automotive condenser (Stevanovic *et al*., 2012) and

Corresponding Author: Pang Suh Chyn, Department of Mechanical Engineering, Faculty of Engineering, University of Malaya, 50603 Kuala Lumpur, Malaysia Tel/Fax: +603-79674448

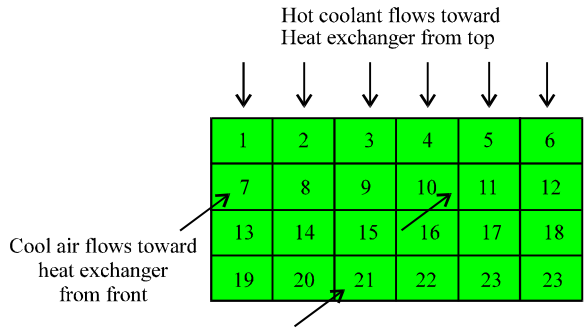

Fig. 1: Segmented heat exchanger with its numbering

three fluids cross-flow heat exchanger (Yuan, 2003). In these studies, the heat exchangers were segmented into elements in order to allow different input parameter (i.e. air flow rates, air inlet temperature). The output parameters such as heat dissipation can be plotted to investigate the deterioration effect of air maldistribution. Finally, basic equations were formulated for modelling heat transfer through heat exchangers (Khaled *et al*., 2011a, b). The heat exchangers were divided into a matrix of 4×4 cells (Fig. 1); heat transfer in each cell will be computed with the basic equations formulated in cell level. The equations take into consideration the energy balance in each of the cell.

METHODS

Thermal equilibrium of coolant circuit: In this research, we would like to describe the thermal equilibrium of engine coolant circuit in equation form, Eq. 1. The engine releases heat as one of the output from fuel combustion. Engine heat transfers through the engine wall into water jacket as there is a temperature gradient (the 1st term is engine heat). The coolant flows inside coolant circuit and dissipates heat to ambient air when the coolant reaches at radiator (the 2nd term is radiator heat). Some negligible heat loss is possible during the frictional coolant flow in the circuit (the 3rd term is heat loss). Finally, the net heat input of engine cooling system is equivalent to summation of the previous three terms (the 4th term is system heat input). The net heat input will be used to increase the coolant temperature (internal energy) or to boil the coolant:

$$Q_{engine,\ out}+Q_{(radiator,\ out)}+Q_{loss}+Q_{input}=0 \qquad (1)$$

After some understanding about thermal equilibrium of engine coolant circuit, we can proceed to perform our mathematics modeling sequentially. In Table 1, it showed the stages taken to perform the full mathematics modeling

Table 1: Step-by-step procedure to obtain engine cooling parameters

Stage 1	Stage 2	Stage 3
Engine speed	Engine speed	Engine speed
Coolant flow	Coolant flow	Coolant flow
Engine heat	Engine heat	Engine heat
	Vehicle speed	Vehicle speed
	Driving gear	Driving gear
		Hood geometry
		Radiator air flow
		Radiator heat
		Dissipation

Table 2: Driving gear and their respective transmission gear ratio

Gear	Transmission gear ratio
1	2.315
2	1.568
3	1.195
4	1.000
5	0.915

and analysis. In Stage 1, the three input parameters are ready available as industry data. In Stage 2, two parameters are computed by correlating engine speed, coolant flow and engine heat with vehicle speed. In Stage 3, we would like to compute the radiator heat which is the second term in Eq. 1. The detailed computation for Stage 2 and 3 will be described thoroughly after this.

The speed of mechanical water pump is proportional to engine speed. So the engine heat and coolant flow increase proportionally with the increment of engine speed. During industry collaboration, we obtained the data of engine speed; coolant flow and engine heat (Stage 1). In Stage 2 we would like to correlate vehicle speed to engine speed, Eq. 2 is listed to perform the calculation. Vehicle speed is a function of engine speed, transmission gear ratio, differential gear ratio and radius of tire. The radius of tire and differential gear ratio are a constant value. However, the transmission gear ratio is varied with driving gear, as tabulated in Table 2:

$$V_{car}=\frac{V_{engine}}{Gr_{trans,i}\times Gr_{diff}}\times 2\times\pi\times r_{tyre}\times\frac{1}{60} \qquad (2)$$

where, i is gear 1, 2, 3, 4, 5 and $0.9< Gr_{trans,\ I} <2.315$. In Stage 3, we aimed to compute radiator heat. The parameter data obtained in Stage 1 and Stage 2 will be fed into a computational fluid dynamics (CFD) model. The engine compartment geometry is also imported into the CFD. The important steps in CFD are geometry import, surface meshing, volume meshing, physics modelling and simulation execution. After we execute the CFD model, radiator air flow and radiator heat could be read from the dual stream heat exchangers. With this, we could perform further analysis for thermal equilibrium of coolant circuit.

Heat exchangers' segmentation: When averaged heat transfer for a heat exchanger is computed from an averaging value of fluids flow rates and fluids temperature differences, significant computational error is incurred. Equation 3 displayed the averaging calculation for heat exchangers' heat dissipation. This is because majority portion of heat exchanger has different air flow rates, different air inlet temperature, different heat dissipation and etc. In short, the heat dissipation is not uniform across the surface area of heat exchanger:

$$\overline{q} = \overline{\dot{m}_{air} C_p}\ \overline{\Delta T_{air}} = \overline{\dot{m}_{coolant}}\ C_p\ \overline{\Delta T_{coolant}} \tag{3}$$

In order to eliminate the error, we formulated equations in order to perform computation in segments level. With these equations, we can apply computation to each and every segment. Different air flow rates, air inlet temperature and other input parameters could be varied for every segment. The output parameters for each segment can be computed with higher accuracy.

In Eq. 4, radiator heat dissipation in segment i can be computed by air flow rate and air temperature rise in segment i. Eq. 5 proven that summation of heat dissipation in each segment is equivalent to total heat dissipation for the heat exchanger. Eq. 6 indicated that coolant inlet temperature for current row is equivalent to coolant outlet temperature for previous row and same column. This equation interconnected the segments. If the heat transfer at previous segment is high and temperature drop is large, the coolant inlet temperature and thus heat transfer for this segment will become low:

$$q_i = \dot{m}_{coolant,i}\ C_{p,\,coolant}\left[\left(T_{coolant,i}\right)_{j,k} - \left(T_{coolant,i}\right)\right] = \dot{m}_{air,i}\ C_{p,air}\left[(T_{air,in})_{j,k} - (T_{air,out})_{j,k}\right]$$
$$\text{where, } j \text{ is row} = 1, 2, 3 \text{ and } 4$$
$$\text{and } k \text{ is column} = 1, 2, 3 \ldots \text{ and } 6 \tag{4}$$

$$q = \sum_{i=1}^{I} q_i \text{ where i segments} = 1, 2 \ldots \ldots 24 \tag{5}$$

$$\left[\left(T_{coolant,\,out}\right)_{j,k} = \left(T_{coolant,\,in}\right)_{j+1,k}\right] \tag{6}$$

The maximum heat to be dissipated in a segment is equivalent to Eq. 7. However, in the context, we need to determine which fluid with a minimum value of (mC_p) is. The fluid with minimum (mC_p) can be considered as "bottleneck fluid" which obstructs further increment in heat transfer. After this, dependent on its thermal effectiveness, the actual heat transfer for segment i can be computed in Eq. 8. The higher the thermal effectiveness, the higher the actual heat transfer is.

$$q_{max,\,i} = (\dot{m}C_p)_{min,i}\,(T_{coolant,\,in,\,i} - T_{air,\,in,\,i}) \tag{7}$$

$$q_i = \varepsilon_i q_{max,\,i} \tag{8}$$

However, the maximum value for thermal effectiveness is one or 100%. After evolution from Eq. 4, Eq. 7 and 8, thermal effectiveness can be formulated in term of fluids temperature in Eq. 9 and 10. Equation 9 applied when the "bottleneck fluid" is coolant and Eq. 10 applied when the "bottleneck fluid" is air. These two equations also add in another two constraints to the problem. One constraint is coolant outlet temperature should be less than or equal to air inlet temperature. Another constraint is air outlet temperature should be less than or equal to coolant inlet temperature:

$$\varepsilon_i = \frac{T_{c,in,i} - T_{c,out,i}}{T_{c,in,i} - T_{a,in,i}} \le 1 \text{ when } (\dot{m}C_p)_{min,i} = (\dot{m}C_p)_{coolant,i} \quad \text{or} \quad T_{c,out,\,i} \le T_{a,in,\,i} \tag{9}$$

$$\varepsilon_i = \frac{T_{a,out,i} - T_{a,in,i}}{T_{c,in,i} - T_{a,\,in,\,i}} \le 1 \text{ when } (\dot{m}C_p)_{min,i} = (\dot{m}C_p)_{air,i} \quad \text{or} \quad T_{a,\,out,i} \le T_{c,in,i} \tag{10}$$

RESULTS

In section 2.1, the concept of thermal equilibrium in engine cooling system is briefed, as in Eq. 1. The stage by stage procedures are followed in order to obtain full set of input parameters and outputs parameters for engine cooling system, as in Table 1. The parameters are tabulated in Table 3. These data will allow us to perform thermal equilibrium analysis of engine cooling system.

Table 3 provided plenty information about engine cooling system. Firstly, it correlated vehicles' speed with engine speed at different driving gear. For instance, the vehicles' can be driven at 60 kph with gear 1, 2 and 3. However, the resulted engine speed and coolant flow and engine heat are varied with gear. For variety vehicle speed and gear combination, how the parameters varied could be observed in the first five columns of Table 3. When the vehicles' driven at lower gear, it could be observed that lower engine speed, lesser engine heat and lower coolant flow.

After execution of CFD model, it can be obtained for the air flow rates and radiator heat dissipation in 6th column and 7th column. Though lesser engine heat is desirable to reduce engine cooling load, however lower coolant flow might incur lower radiator heat dissipation. For instance 60 kph driving with gear 2, 3 and 4, the

Table 3: Computation of parameters required for thermal equilibrium analysis

Vehicle speed (kph)	Gear	Engine Speed (RPM)	Coolant flow rates (kg sec^{-1})	Engine heat (kW)	Air flow rates(kg sec^{-1})	Radiator heat dissipation (kW)
40	1	3358	0.955	15.99	0.209	--
40	2	2272	0.662	12.44	0.209	7.06
50	1	4194	1.155	19.89	0.275	--
50	2	2841	0.805	14.42	0.275	10.26
50	3	2165	0.630	11.99	0.275	9.877
60	1	5034	1.302	23.76	0.342	--
60	2	3409	0.971	16.05	0.342	13.39
60	3	2599	0.752	13.70	0.342	12.83
60	4	2173	0.630	11.99	0.342	12.40
80	2	4545	1.218	21.51	0.477	--
80	3	3465	0.989	16.20	0.477	18.30
80	4	2899	0.818	14.60	0.477	--
110	3	4762	1.260	22.51	0.685	--
110	4	3956	1.109	18.76	0.685	24.72
110	5	3649	1.036	17.10	0.685	--

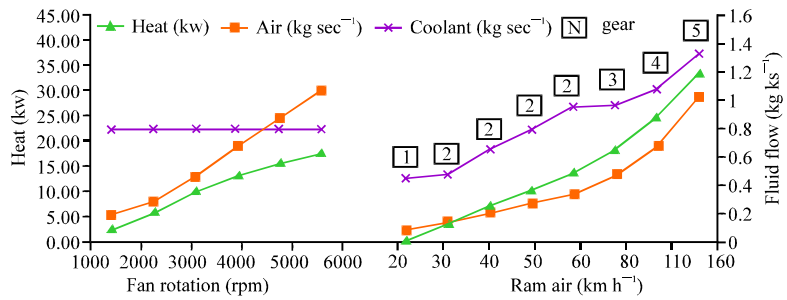

Fig. 2: Radiator heat dissipation at different fan speed and ram air speed

engine heat are 16.05, 13.7 and 11.99 kW; the radiator heat are 13.39, 12.83 and 12.40kW. In order to conclude if higher gear will reduce net heat into engine cooling system, we have to calculate net heat using Eq. 11. The net heat are 2.66, 0.87, -0.41 kW, respectively. The net heat input decreases as driving gear increases, thus can reduce cooling load.

Alternatively, after computation of net heat values, we can calculate how fast the fan speed should be in order to reach thermal equilibrium when $Q_{net} = 0$. The fan speed required to reach thermal equilibrium is lower for those cases with higher driving gear at a similar vehicle speed (50 and 60 kph). For instance vehicle speed at 60 kph, the fan speed required to reach thermal equilibrium are 1056, 351 and 0 rpm, respectively. The fan speed required to reach thermal equilibrium is higher at lower vehicle speed (10 kph). This is because ram air supply (due to vehicle motion) is limited at low vehicle speed. The operation of cooling fan is required when the vehicle speed reaches less than 50-60 kph:

$$Q_{net} = Q_{(engine, out)} + Q_{(radiator, out)} \quad (11)$$

In Fig. 2, the radiator heat dissipation, coolant flow and air flow is plotted against different vehicle speeds at a particular driving gear (for both the fan and ram air cases). It could be observed that radiator heat dissipation increases with fan speed and ram air speed. However, it is noticed that the radiator heat dissipation in the ram air cases is higher than those in fan cases.

This phenomenon could be explained by the CFD contour plot, as shown in Fig. 3. CFD is 3D simulation software and it could capture dynamic air flow approaching radiator. Vehicle motion is pushing the air from front of heat exchangers (ram air) while cooling fan is sucking the air from rear of heat exchangers. In Fig. 3 a, it could be observed that the air flow distribution is evenly distributed for ram air case. While for cooling fan case, high air flow is concentrated at the position of cooling fan only, Fig. 3b. The green circle area experiences the strongest suction force. Primarily, we would like to explain lower radiator heat in cooling fan cases is due to majority frontal surface does not participate in heat dissipation. In order to perform a fair comparison, further analysis is carried out with a fixed coolant rate of 0.79 kg sec^{-1}. In Fig. 4, it could be showed the quantitative differences for radiator heat dissipated by ram air and fan (at a constant air flow and coolant flow).

This also provokes us to develop a spread sheet model to divide the heat exchangers into segments. The segmentation method described in section 2.2 allows us to study the effect of air flow distribution towards the

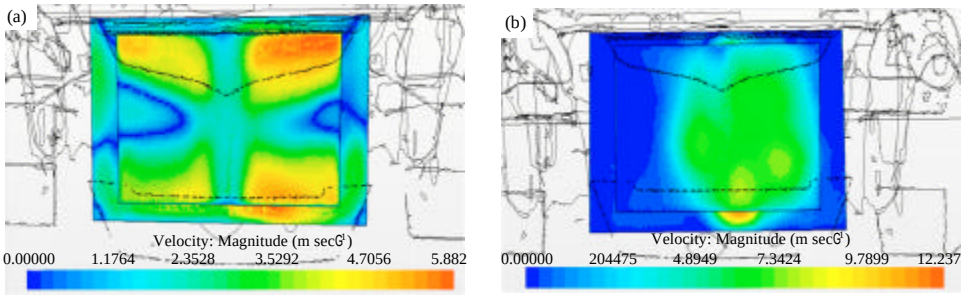

Fig. 3: Air Flow Distribution at Frontal surface of Heat Exchanger for Ram Air and Cooling Fan Scenarios.

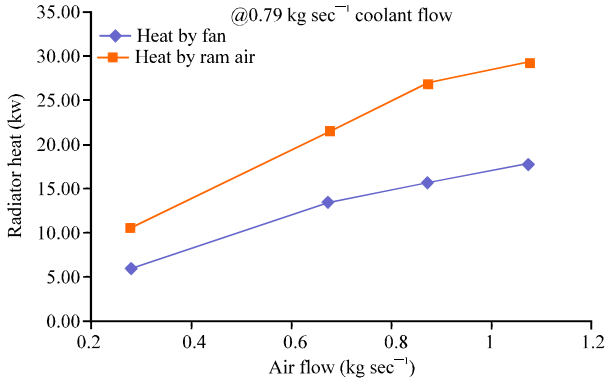

Fig. 4: Quantitative differences of radiator heat dissipation

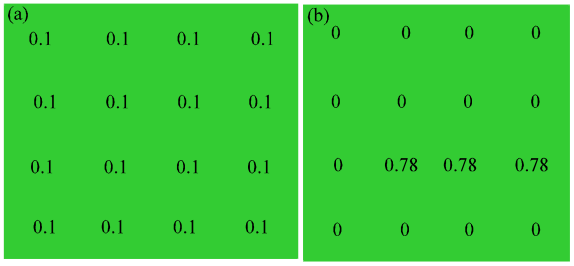

Fig. 5: Segmented Heat Exchangers (a) Evenly Distributed Air Flow (b) High Air Flow Concentrated at Small Area

performance of heat exchanger. Segmentation permits us to set different air flow rates and air temperature for each segment.

In Fig. 5, we utilize the spread sheet model to explain the phenomenon that fan suction dissipates less heat at a similar air flow rate. In the cooling fan cases, the full potential of high air flow is not exploited correctly. Since the heat capacity for air is 1/4 of the heat capacity for coolant, air side convection is much higher in thermal resistance. Air is always the fluid with minimum mC_p or is the "bottleneck fluid" which obstructs further increment in heat dissipation. It is rather wasteful when the potential of "bottleneck fluid' is not exploited correctly.

In Fig. 5a, we determine the air flow rate to be distributed evenly for all segments. The air is the fluid with minimum mC_p in each and every segment. In Fig. 5 b, we determine the air flow to be concentrated in three segments only. The coolant is the fluid with minimum m C_p in the three segments, which means the "bottleneck fluid" now is coolant instead of air. Though high air flow rates through the three segments, however the coolant heat capacity limited total heat dissipation. In short, some air does not participate in heat dissipation.

CONCLUSION

In early stage of research, industry data of engine heat and coolant flow at a particular engine speed were obtained and analyzed. In order to visualize the engine heat dissipated at a particular speed, the said parameters were correlated with vehicle speed and driving gear. Later, the CFD model provided the contour result of air flow distribution at frontal face for heat exchanger. CFD model also captured the air flow throughput at heat exchanger. In the dual stream heat exchanger report, heat dissipated at radiator was summarized. After the engine heat and radiator heat dissipation were obtained, the thermal equilibrium of engine cooling system was discussed. If the engine heat is greater than radiator, it was presumed that an amount of heat input into engine cooling system. The heat input could be stored as internal energy of coolant (temperature increase) or be used to boil the coolant. It was found out that driving a vehicle speed at higher gear could help to reduce cooling load. Through the thermal equilibrium study, we could calculate when the fan interference is required and what fan speed is required. In the CFD model, it was discovered that radiator heat dissipation is higher for the uniform ram air cases

than cooling fan cases. A spreadsheet model was established to explain this phenomenon. In the spreadsheet model, the heat exchanger is segmented to 24 portions in order to study the effect of air flow distribution. When high air flow concentrates at a small portion of area, this permits the fluid with minimum mC_p to be switched from air to coolant. When coolant is the "bottleneck fluid", any increment in air (the non-bottleneck fluid) will not provoke increment in heat dissipation.

ACKNOWLEDGEMENT

I would like express my gratitude and appreciation towards University of Malaya and Malaysia Ministry of Higher Education (MOHE) for providing research funding to our research team (Project: UM.C/HIR/MOHE/ENG/60).

REFERENCES

Agrawal, M., N. Adlakha and K.R. Pardasani, 2010. Three dimensional finite element model to study heat flow in dermal regions of elliptical and tapered shape human limbs. Applied Math. Comput., 217: 4129-4140.

Giraldo, M., D. Sanin and W.F. Florez, 2011. Heat transfer in nanofluids: A computational evaluation of the effects of particle motion. Applied Math. Comput., 219: 3308-3315.

Habib, M.A., R. Ben-Mansor, S.A.M. Said, M.S. Al-Qahtani, J.J. Al-Bagawi and K.M. Al-Mansour, 2009. Evaluation of flow maldistribution in air-cooled heat exchangers. Comput. Fluids, 38: 677-690.

Hoffmann-Vocke, J., J. Neale and M. Walmsley, 2011. The effect of inlet conditions on the air side hydraulic resistance and flow maldistribution in industrial air heaters. Int. J. Heat Fluid Flow, 32: 834-845.

Kaern, M.R., W. Brix, B. Elmegaard and L.F.S. Larsen, 2011. Performance of residential air-conditioning systems with flow maldistribution in fin-and-tube evaporators. Int. J. Refriger., 34: 696-706.

Kaneko, H., K.S. Bey, Y. Lenbury and P. Toghaw, 2008. Numerical experiments using hierarchical finite element method for nonlinear heat conduction in plates. Applied Math. Comput., 201: 414-430.

Kaveh, A. and M.J. Tolou Kian, 2012. Efficient finite element analysis using graph-theoretical force method with brick elements. Finite Elements Anal. Des., 54: 1-15.

Khaled, M., F. Harambat, H. El Hage and H. Peerhossaini, 2011a. Spatial optimization of an underhood cooling module-Towards an innovative control approach. Applied Energy, 88: 3841-3849.

Khaled, M., F. Harambat and H. Peerhossaini, 2011b. Analytical and empirical determination of thermal performance of louvered heat exchanger: Effects of air flow statistics. Int. J. Heat Mass Transfer, 54: 356-365.

Li, C.H., H.S. Chiou, C. Hung, Y.Y. Chang and C.C. Yen, 2002. Integration of finite element analysis and optimum design on gear systems. Finite Elements Anal. Des., 38: 179-192.

Maity, D., 2005. A novel far-boundary condition for the finite element analysis of infinite reservoir. Applied Math. Comput., 170: 1314-1328.

Stevanovic, V., S. Cucuz, W. Carl-Meissner, B. Maslovaric and S. Prica, 2012. A numerical investigation of the refrigerant maldistribution from a header towards parallel channels in an evaporator of automotive air conditioning system. Int. J. Heat Mass Transfer, 55: 3335-3343.

Yuan, P., 2003. Effect of inlet flow maldistribution on the thermal performance of a three-fluid crossflow heat exchanger. Int. J. Heat Mass Transfer, 46: 3777-3787.