地面效應對汽車冷卻氣流分佈的影響

楊子為、黃國修

E-mail: 9419913@mail.dyu.edu.tw

摘 要

汽車行駛過程中引擎室內會產生散熱問題，在引擎熱量管理及冷卻系統分析領域上，引擎室前端流入足夠冷卻流量對水箱冷卻系統而言，將直接有效維持引擎效率表現。因此引擎室冷卻系統分析中，流過至水箱之空氣質量流率對冷卻系統效率為一項具關鍵性參數的影響，與汽車燃油經濟性具有直接性關係。 本研究採用計算流體力學(CFD)，反應汽車於路面行駛中之真實物理現象，加入地面效應之距地高度、地面移動及車輪轉動參數探討對汽車冷卻氣流分佈的影響。分析汽車行駛狀態於怠速、50km/hr、110km/hr及180km/hr時，地面效應參數對引擎室水箱空氣質量流率、入口端及出口端流場分佈的影響作深入探討。 結果指出距地高度為影響引擎室水箱空氣質量流率之關鍵參數，怠速時比對最小距地高度與最大距地高度之水箱空氣質量流率差為8%，隨行駛速度提升至180km/hr時兩者距地高度於水箱空氣質量流率差為15%。地面移動與車輪轉動參數於三者不同距地高度數值分析後，對於水箱空氣質量流率無太大差異，影響程度可以忽略。 採用汽車尺寸模型比對此地面效應參數影響，50km/hr全車尺寸模型、半車尺寸模型與車頭尺寸模型比對水空氣質量流率誤差在1%，180km/hr誤差為3%，符合文獻流場趨勢[12]。

關鍵詞 : 距地高度 ; 地面效應 ; 冷卻氣流

目錄

參考文獻

[1] 李添財，汽車空氣動力學，全華科技圖書股份有限公司，民國80，第2-15頁、第4-5 頁。

[2] Habchi, S., HO, S. D., Elder, J. and Singh, S.,“ Airflow and Thermal Analysis of Underhood Engine Enclosures, ” SAE Technical Paper 1994-01-0316, 1994.

[3] Ohshima, T., Hamatani, K., Ninoyu, M. and Nakagawa, K.,“ Influence of the Cooling Air Flow Outlet on the Aerodynamic Characteristics, ” JSAE Review pp. 137-142, 1998.

[4] Williams, J. and Vemaganti, G.,“ CFD Quality – A Calibration Study for Front – End Cooling Airflow, ” SAE Technical Paper 980039, 1998.

[5] Ono, K., Fujitani, K. and Fujita, H.,“ Applications of CFD Using Voxel Modeling to Vehicle Development, ” ASME Paper fedsm99-7323, 3rd ASME/JSME Fluids Engineering Conference, San Francisco, California, July 18-23, 1999.

[6] Srun, N.,“ A Simple Engine Cooling System Simulation Model, ” SAE Technical Paper 1999-01-0237, SAE International Congress & Exposition, Detroit, Michigan, March. 1-4, 1999.

[7] Lawrence, V.,“ Underhood Airflow Simulation of a Passenger Car Using Computational Fluid Dynamics, ” SAE Technical Paper 2001-01-3800, 2001.

[8] Viviam, L.,“ Underhood Airflow Simulation of a Passenger Car Using Computational Fluid Dynamics ” SAE Technical Paper 2001-01-3800,

2001.
[9] Chang, F. C., Malipeddi, S. R., Uppuluri, S. and Shapino, S., " Underhood Thermal Management of Off-Highway Machines Using 1D-Network Simulations, " SAE Technical Paper 2003-01-3405, 2003.
[10] Yang, Z., Bozeman, J., Shen, F. Z. and Acre, J. A., " CFRM Concept at Vehicle Idle Conditions, " SAE Technical Paper 2003-01-0613, 2003.
[11] Williams, J., " Aerodynamic Drag of Engine-Cooling Airflow with External Interference, " SAE Technical Paper 2003-01-0996, 2003.
[12] Huang, K. D. and Tzeng, S. C., " Optimization of Size of Vehicle and Flow Domain for Underhood Airflow Simulation, " Proceedings of the I MECH E Part D Journal of Automobile Engineering, vol. 218, no. 9, pp. 945-951(7), 1 September 2004.
[13] Ng, E.Y., Watkins, S. and Johnson, P. W., " New Pressure-Based Methods for Quantifying Radiator Airflow, " Proceedings of the I MECH E Part D Journal of Automobile Engineering, vol. 218, no. 4, pp. 361-372(12), 1 April 2004.
[14] Wiedemann, J., " The Influence of Ground Simulation and Wheel Rotation on Aerodynamic Drag Optimization-Potential for Reducing Fuel Consumption, " SAE Technical Paper 960672, 1996.
[15] Aronson, D., Perzon, S. and Brahim, S. B., " On the Underbody Flow of a Simplified Estate, " SAE Technical Paper 2000-01-0485, 2000.
[16] Skea, A. F., Bullen, P. B. and Qiao, J., " CFD Simulations and Experimental Measurements of the Flow Over a Rotating Wheel in a Wheel Arch, " SAE Technical Paper 2000-01-0487, 2000.
[17] FLUENT Inc., " Flow Around an Automobile Wheel, " Journal Articles by FLUENT Software Users, EX193, pp. 1-2, 2002.
[18] Kleber, D. I. A., " Simulation of Air Flow Around an OPEL ASTRA Vehicle with FLUENT, " Journal Articles by FLUENT Software Users, JA132, pp. 1-6, 2001.
[19] Elofsson, P. and Bannister, M., " Drag Reduction Mechanisms Due to Moving Ground and Wheel Rotation in Passenger Cars, " SAE Technical Paper 2002-01-0531, 2002.
[20] Launder, B. E. and Spalding, D. B., " Lectures in Mathematica1 Model of Turbulence, " Academic Press, 1972.
[21] http://www.theautochannel.com/newcardb/cccars.html?bodystyle =All%20Sedans&titleid=326 [22] Fluent 6.1 Manuals, Fluent Incorporation.