Evaluating the performance of a water-cooled scooter engine through a 3-D numerical and experimental analysis

Hao-Pin Lien\(^2\), Jiin-Yuh Jang\(^1\) and Por-Kang Chen\(^1\)

1 Department of Mechanical Engineering, National Cheng Kung University, No.1, University Road, Tainan, Taiwan, 70101.
2 Kwang Yang Motor Co., Ltd. Kaohsiung, Taiwan, 80794

Abstract. In this study, a 3-D numerical model of a 400c.c four-stroke single-cylinder, water-cooled scooter engine is established and evaluated through computational fluid dynamics (CFD) with a conjugated heat transfer scheme to show the temperature distribution of the engine. The flow field of the water jacket is also observed, and the correlation between the heat transfer performance and flow field is discussed. In order to observe the real coolant flowing through the inside of the water jacket and make a comparison with the numerical results, we establish a transparent water jacket model using a 3-D printer. For the sake of developing the water jacket guidelines, we arrange the number of the gasket holes to acquire different flow fields and thermal results.

1. Introduction
Without a doubt, the temperature of internal combustion engines plays a crucial role in engine performance and endurance. If the temperature goes too high, it causes damage to the combustion chamber, the cylinder liner, the valves, the valve seats, and the plug seat. On the other hand, if the temperature goes too low, it decreases the efficiency of lubrication and combustion. Hence, keeping engines operating under adequate temperature conditions is a critical issue that needs to be addressed.

2. Theoretical Model
In this paper, a 3-D numerical model for a 400c.c four stroke single cylinder water-cooled scooter engine was established, as shown in Figure 1. The yellow region is the cooling jacket; the black region is the gasket, and the other region is the engine parts. Different gaskets are shown in Figure 2. Case1 is the original design from the scooter manufacturer. In order to optimize the flow field in the water jacket, the distribution of holes was rearranged in different ways on the gasket, as shown in cases 2 to 12.

Figure 1. The layout of engine model
2.1. Governing Equations

The mass conservation equation is given by:
\[
\frac{\partial \rho_j}{\partial t} + \nabla \cdot (\rho u_j) = 0
\]  
(1)

The momentum equation is given by:
\[
\rho \frac{\partial \mathbf{u}}{\partial t} = \nabla \cdot (-\rho \mathbf{u} \mathbf{u} - p \mathbf{I} + \tau) + \rho f
\]
\[
\nabla \cdot \tau = \mu \left( \nabla \mathbf{u} + (\nabla \mathbf{u})^T \right) - \nabla p
\]
(2)

The energy equation related to the fluid part (coolant water) is given by:
\[
\rho C_p \frac{\partial T}{\partial t} = \nabla \cdot (\rho C_p \mathbf{u} T) + \nabla \cdot (k \nabla T) - \rho C_p \frac{\partial}{\partial x_j} \left( u_j' T' \right)
\]
(3)

For the solid part (engine), the heat conduction equation may be expressed as a Laplace equation:
\[
\nabla^2 T = 0
\]
(4)

3. Experimental Set-Up

In order to observe the flow field inside the engine, the experimental set-up is shown in Figure 3. It consists of a water pump, a Karman vortex flow meter, and a transparent 3-D printed water jacket model, which is shown in Figure 3. The transparent 3-D printed water jacket model contains two parts, a cylinder side and a cylinder head side, in order to separate and readily adjust different hole arrangements. We used a sharp tool to scarify the holes on the mat and change the distributions of these holes with different cases as mentioned above. These flow rate and velocity values are shown in Table 1.

(continued on page 3)
4. Results and Discussion

4.1. Model Validation
In order to verify the simulation, the real experimental data should be compared with the numerical results. The experimental data for Case 1 and Case 2 were provided by the scooter manufacturer. The locations for the temperature sensors are indicated in Figure 4. For both cases, there are 13 sensors allocated on the cylinder head and 4 sensors allocated on the cylinder. It is shown in Table 2 that the calculated temperature data were in good agreement with the experimental data within a 10% difference.

![Figure 4. Location of thermal couples.](image)

| Position | Experimental | Simulation | Error |
|----------|--------------|------------|-------|
| T1 | 115.6 | 117.2 | 1.41 |
| T2 | 105.9 | 107.9 | 1.91 |
| T3 | 111.9 | 113.4 | 1.28 |
| T4 | 128.6 | 134.6 | 4.59 |
| T5 | 144 | 154.6 | 7.56 |
| T6 | 91.5 | 93.8 | 2.78 |
| T7 | 91.5 | 94.4 | 3.75 |
| T8 | 91.5 | 94.4 | 3.75 |
| T9 | 91.5 | 94.4 | 3.75 |
| T10 | 91.5 | 94.4 | 3.75 |

Table 2. Difference between experiment and simulation of case 1 and case 2.

4.2. Flow Field for Different Gasket Openings
The total water volume flow rate and the percentage of water passing through the open holes for the different gasket openings are shown in Table 3. As a result, the gasket hole closer to the water jacket exit will cause more coolant to pass through. Figure 5 shows that the flow rates in Case 4, Case 5, and Case 6 are less in the cylinder-head side of the water jackets as compared with the flow rates in Case 1, Case 2, and Case 3. Figure 8 also shows that more holes are opened with the exception of Case 8. It can be observed that these cases where the flow rates passing through hole No. 4 to hole No. 8 to the exit are lower than for other holes, especially in Case 9, where approximately 44% of the total coolant flows through hole No. 6, which is closest to the exit. This phenomenon concurs with the previous cases discussed above, which caused deficiencies in the heat coefficient. In Figure 5, we can see the side view of the real flow field in the water jacket and experimental flow field. For the experimental flow field, it is similar to the simulated streamline depicted in Figure 5. Using this method, we can validate the numerical results.
4.3. Heat Transfer Coefficient for Different Gasket Openings
The heat transfer coefficient distribution and average heat transfer coefficient for different gasket openings are shown as Figure 7 and Table. 4. However, the corresponding average heat transfer coefficients are reduced from 3 to 11%. The result shows that the heat transfer coefficient for Case 2 is increased around 4.2% more than other cases. There are many important components located at the cylinder head such as the exhaust valves, intake valves, and valve seats. Therefore, the heat transfer efficiency of the cylinder head should be taken into account. As the value of the convection coefficient $h$ becomes lower, a high temperature distribution will result.

| Table 4. Average heat transfer coefficient for different gasket openings. |
|---|
| **Case** | **Case 1** | **Case 2** | **Case 3** | **Case 4** | **Case 5** | **Case 6** | **Case 7** | **Case 8** | **Case 9** |
| **Flow rate (%)** | 1.2 | 2.3 | 3.5 | 4.6 | 5.7 | 6.8 | 7.9 | 8.1 | 9.2 |
| **Case 10** | 10.3 | 11.4 | 12.5 | 13.6 | 14.7 | 15.8 | 16.9 | 17.0 | 18.1 |
| **Case 11** | 19.2 | 20.3 | 21.4 | 22.5 | 23.6 | 24.7 | 25.8 | 26.9 | 28.0 |
| **Case 12** | 29.1 | 30.2 | 31.3 | 32.4 | 33.5 | 34.6 | 35.7 | 36.8 | 37.9 |
| **Average** | 14.45 | 15.56 | 16.67 | 17.78 | 18.89 | 19.90 | 20.91 | 21.92 | 22.93 |
| **Standard Deviation** | 1.23 | 1.34 | 1.45 | 1.56 | 1.67 | 1.78 | 1.89 | 1.90 | 1.91 |
4.4. Cylinder Head Temperature for Different Gasket Openings
For all cases, the highest temperature region existed between the two exhaust valve seats in the combustion chamber, as shown in Figure 8. Compared with all of the other cases, there was the most homogeneous and highest heat transfer coefficient around the exhaust side in Case 2. Therefore, the temperature could be lower than in the other cases.

4.5. Different Arrangements of Gasket Holes Cause Various Mean Temperature on the Exhaust Valve Seat
Table 5 shows the different mean temperatures on the exhaust valve seat because of different arrangements of the gasket holes. The locations of the exhaust valve seats are depicted in Figure 9. The seat closest to the plug is marked in red, and the one closest to the valve chain is marked in blue. The region where the exhaust valves and valve seats are touched is shown in yellow. Table 5 shows that the temperature for Case 2 on the plug side is not the lowest; the temperature on the chain side is the lowest. Also, the value of the difference between the plug and chain temperature is only 0.3 degrees, which is the smallest in all Cases.
Table 5: Exhaust valve seat mean temperature.

| Case | \(T_{avg} \text{ °C} \) | \(T_{max} \text{ °C} \) | \(T_{avg} - T_{max} \text{ °C} \) |
|------|-----------------|-----------------|-----------------|
| Case 1 | 299.6 | 293.7 | 4.1 |
| Case 2 | 299.1 | 290.4 | 0.3 |
| Case 3 | 299.7 | 294.5 | 4.8 |
| Case 4 | 297.7 | 296.1 | 1.6 |
| Case 5 | 294.8 | 294.1 | 0.1 |
| Case 6 | 298.2 | 295.8 | 2.4 |
| Case 7 | 296.3 | 297.1 | 0.8 |
| Case 8 | 298.8 | 293.7 | 4.9 |
| Case 9 | 330.8 | 333.7 | 2.9 |
| Case 10 | 291.5 | 296.4 | 4.9 |
| Case 11 | 297.1 | 301.7 | 4.6 |
| Case 12 | 292.1 | 294.7 | 2.6 |

5. Conclusion

According to the results, there are some valuable conclusions as addressed below:

1. Through the 3-D printed water jacket model, we were able to observe the actual streamline in the water jacket. In addition, this result could be compared with the numerical flow field results for the sake of validation of the numerical simulation.

2. As the number of holes on the gasket was increased from 2 to 8, the flow rates increased by as much as 20.2%. After the coolant flowed into the water jacket, it was divided into two parts. One part of the fluid entered the cylinder head in a clockwise direction, while the other part entered the cylinder head in a counter-clockwise direction. We found that the coolant flow concentrated through the hole closest to the exit. For this reason, setting the hole close to the exit had a negative effect on performance.

3. Compared to the original design, which was Case 1, the flow rate for Case 2 increased by 7.14%, and the average heat transfer coefficient increased by around 4.25%. For cases 7 to 12, the flow rates increased by 16 to 20%, while the average heat transfer coefficients reduced by around 5.1 to 11.7%. As a result, it could be inferred that the flow rate does not have direct correlation with the heat transfer coefficient, but the flow field does.

4. The highest temperature region exists between the two exhaust valve seats in a combustion chamber. Case 2 exhibited the lowest and most uniform temperature distribution around the region of the exhaust valve seats.

5. To obtain high performance of a water jacket in a water-cooled engine, the gasket holes should be arranged away from the exit.

Acknowledgements

This research was financially supported by the Ministry of Science and Technology, Taiwan, under contracts MOST 106-2622-8-006-001.

References

[1] Agarwal, D et al. (2015). Heat Transfer Analysis of a Medium Duty DI Diesel Engine. International Journal of Informative & Futuristic Research Vol. 2, No. 10, 3627-3637

[2] ANSYS Fluent (2013). A Release 15.0 Documentation, ANSYS, Inc.

[3] Deger, Y et al. (2004). Coupled CFD-FE-Analysis for the Exhaust Manifold of a Diesel Engine. ABAQUS Users' Conference, 199-208.

[4] Fontanesi, S. and Giacopini, M. (2013). Multiphase CFD–CHT optimization of the cooling jacket and FEM analysis of the engine head of a V6 diesel engine. Applied Thermal Engineering Vol. 52, No. 2, 293-303

[5] Jafarabadi, Met al. (2008). A Fast Coupled Cfd-Thermal Analysis of a Heavy Duty Diesel Engine Water Cooling System. ASME International Mechanical Engineering Congress and Exposition, pp.1-10
[6] Jang et al. (2010). 3-D Quasi-Steady State Heat Transfer Analysis for an Air-Cooled Motorcycle Engine. *Advanced Materials Research, Vol. 129-131*, 1322-1325

[7] Lin, W.T (2015). Three Dimensional Thermal Hydraulic Analysis for Water-Cooled Motorcycle Engine. Thesis of Master from National Cheng Kung University. Advisor: Jang, J. Y.

[8] Mulemane, A. and Soman, R. (2007). CFD Based Complete Engine Cooling Jacket Development and Analysis. *SAE Technical Paper 2007-01-4129*, pp1-10

[9] Wu, M.H. (2015). Analysis of Engine Cylinder Heat Distribution and Combustion. Research Report from Kwang Yang Motor Co. Ltd.