COMPUTATIONAL FLOW MODELLING OF FORMULA-SAE SIDEPODS FOR OPTIMUM RADIATOR HEAT MANAGEMENT

C. M. DE SILVA*, M. NOR AZMI, T. CHRISTIE, E. ABOU-SABA, A. OOI

Department of Mechanical Engineering,
The University of Melbourne, Victoria 3010, Australia
*Corresponding Author: c.desilva3@pgrad.unimelb.edu.au

Abstract
Formula SAE vehicles, over the program’s history have showcased a myriad of aerodynamic packages, each claiming specific quantitative and qualitative features. This paper attempts to critique differing aerodynamic sidepod designs and their effect upon radiator heat management. Various features from inlet size, sidepod shape and size, presence of an undertray, suspension cover, gills and chimneys are analysed for their effects. Computational Fluid Dynamics (CFD) analyses are performed in the FLUENT environment, with the aid of GAMBIT meshing software and SolidWorks modelling.

Keywords: Heat transfer, Downforce, Drag, SAE.

1. Introduction
The Formula Society of Automotive Engineers – Australasia (FSAE-A) competition is the Australasian arm of the international event run by the Society of Automotive Engineers, where teams of students are challenged to conceive, design and fabricate a small formula style race car. Since 2005, engine overheating has been a primary concern for the Melbourne University Racing (MUR) team at the Formula SAE-A Endurance Event, which has lead to both performance and reliability issues. The engine cooling systems are critical for the car to run smoothly, a failure in the cooling system would mean a failure in the overall car. Use of a cooling system with excess capacity does not provide a solution either as this adds unnecessary cost and weight. Thus, the design of an engine cooling system is a task in optimization [1].

Many factors contribute to overheating; the most critical is the inadequate management of airflow through the radiator [1]. As the FSAE-A event is held during
Nomenclatures

\[ C_D \quad \text{Coefficient of Drag} \]
\[ C_L \quad \text{Coefficient of lift} \]
\[ \varepsilon \quad \text{Epsilon-NTU method} \]
\[ F_D \quad \text{Drag force, N} \]
\[ F_L \quad \text{Lift force, N} \]
\[ h \quad \text{Heat transfer coefficient} \]
\[ k_L \quad \text{Pressure loss coefficient} \]
\[ m \quad \text{Mass flow rate of air, kg/s} \]
\[ \Delta p_F \quad \text{Pressure rise across radiator fan, Pa} \]
\[ \Delta p_R \quad \text{Pressure drop across radiator, Pa} \]
\[ Q \quad \text{Heat transfer, W} \]
\[ v \quad \text{Sidepod inlet velocity, m/s} \]
\[ v_s \quad \text{Swirl velocity, m/s} \]
\[ r_n \quad \text{Polynomial constant} \]
\[ y^+ \quad \text{Dimensionless wall distance} \]

Greek Symbols

\[ \alpha \quad \text{Angle of attack, deg.} \]

during the summer months, temperatures on the track can rise considerably. Previous MUR teams have fallen victim to this overheating issue at FSAE-A competitions. In 2008, alone, two engines were damaged as a result of poor heat management. Consequently, previous MUR drivers have been unable to perform competitively during the endurance event for fear of overheating and damaging an engine. Critical variables that may be manipulated to solve heat management issues include radiator size, airflow through the radiator and water flow rate through the radiator. It is widely accepted that airflow rate plays a major role in providing heat transfer. This is evident in Fig. 1 [2].

Figure 1 shows the heat transfer rate, \( Q \), from the radiator which is heavily influenced by the radiator dimensions and the air velocity through the radiator.

![Fig. 1. Heat Dissipated, \( Q \), for Different Airflow Rates and Radiator Dimensions [2].](image)

*Journal of Engineering Science and Technology*  
February 2011, Vol. 6(1)
These results were obtained by [2] by modelling the radiator using analytical calculations together with experimental correlations, as it is not possible to directly calculate the heat transfer of a given radiator core. The radiator was modelled as a crossflow heat exchanger using the ε-NTU method together with all assumptions associated with this method [3]. There is a discontinuity in the graph due to the transition from laminar to turbulent, in the water flow within the radiator fins. Whilst the choice of radiator dimensions are made by the 2009 MUR Engine sub team [2], it is a requirement of the 2009 MUR Aerodynamics and Safety sub team to optimize the airflow rate through the radiator.

2. CFD Analysis

2.1. Modelling the radiator and the fan

- **Lumped-parameter model**

  The cooling subsystem of the car can be designed using CFD analysis. However, modelling the radiator and fan in CFD is difficult due to their complex geometry. Fortunately, a lumped-parameter model for a radiator and a fan is available in FLUENT. Though more complex heat exchanger models are available, the lumped-parameter model is sufficient. This is because the CFD analyses conducted are only to understand the impact of the existence of the radiator and the fan in the flow field within the sidepod. The radiator model is simplified in the analysis as a face within the GAMBIT environment with the appropriate boundary conditions selected, this is necessary to use the lumped parameter model for the radiator as it requires the radiator to be modelled as an infinitely thin entity [4].

- **Pressure loss coefficient across radiator**

  The pressure loss coefficient, \( k_L \), is necessary for the lumped-parameter model in FLUENT. The pressure drop through the radiator is assumed to be proportional to the dynamic head of the fluid, with an empirically determined loss coefficient \( k_L \) [4], which has the general form given in Eq. (1),

  \[
  k_L = \sum_{n=1}^{N} r_n v^{n-1}
  \]

  where \( r_n \) are polynomial constants and \( v \) is the magnitude of the local fluid velocity normal to the radiator, in this case the inlet velocity. Figure 2 shows a plot of \( k_L \), against the air velocity through the radiator core obtained experimentally.

  The pressure drop \( \Delta p_R \) across the radiator was determined for different inlet velocities \( v \), using Pitot-tubes placed before and after the radiator. This was then used in Eq. (2) to determine \( k_L \). From the plot, \( k_L \) can be written as a polynomial function of the velocity through the radiator core, as given in Eq. (3). This polynomial is then used in the FLUENT radiator model.

  \[
  \Delta p_R = k_L \frac{1}{2} \rho v^2
  \]

  \[
  k_L = 0.286v^4 - 4.257v^3 + 22.79v^2 - 53.07v + 50.66
  \]
• Heat loss coefficients, \( h \)

The heat transfer through the radiator core can be modelled by measuring the heat transfer coefficient, \( h \), at various inlet velocities, \( v \), in an experiment. From [5], the polynomial relationship of the heat transfer coefficient through the radiator core is given by Eq. (4). This polynomial relationship was determined by experimental investigations performed by the 2006 MUR aerodynamics team, where heat transfer from the radiator was determined for varying inlet velocities set by the wind tunnel. This polynomial is also necessary for the lumped parameter model in FLUENT to model the heat transfer across the radiator.

\[
h = 352.51v + 580.52
\]  

(4)

• Pressure rise across the fan

The fan model can be used to determine the impact of a fan with known characteristics upon some larger flow field. Some of the characteristics of the fan need to be determined to set its boundary condition. In FLUENT, the pressure rise across the fan and the swirl velocity can be set. The pressure rise across the fan is dependent on the air velocity and is given by Eq. (5),

\[
\Delta p_f = -0.37v_s^3 + 4.49v_s^2 - 40.06v_s + 294.9
\]  

(5)

where \( \Delta p_f \) is the pressure rise across the fan and \( v_s \) is the swirl velocity. This equation was determined by fitting a polynomial to manufacturer data given for pressure drop for varying swirl velocities in [6]. The lumped parameter model for the fan does not provide an accurate description of the detailed flow through the fan blades. Instead, it predicts the amount of flow through the fan which is what we are interested in for this study.
2.2. CFD validation

To validate the CFD results, comparisons were drawn between obtained results and published experimental data [7]. For this purpose, the NACA 0018 model was analysed for several angles of attack, $\alpha$, ranging from 0º to 25º. Two different types of meshes were implemented to explore the relationship between the structure of the mesh and the accuracy of results. The coarse mesh consisted of $y^+ < 100$ at the surface of the airfoil compared with the finer mesh which had $y^+ < 5$. The iteration time for the coarse mesh is considerably lower than the fine mesh, as shown in Table 1. However, the fine mesh provides more accurate results for $C_L$ compared to the coarse mesh, especially for higher $\alpha$ (Fig. 3). Coarse mesh results are within 30% error, whereas for the fine mesh the maximum error is around 5%. The inaccuracy in results is primarily caused by the fact that at higher angles of attack, more turbulence is evident downstream, hence it is evident that more elements (a finer mesh) is necessary to capture the rapid variations in the static pressure to give a more accurate $C_L$.

Table 1. Iteration Time and Number of Elements for NACA 0018 at $\alpha = 20^\circ$.

<table>
<thead>
<tr>
<th>Mesh Type</th>
<th>Time per 100 iterations (min)</th>
<th>Number of elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>0.9</td>
<td>50,000</td>
</tr>
<tr>
<td>Fine</td>
<td>24.8</td>
<td>600,000</td>
</tr>
</tbody>
</table>

Figure 3. Lift Coefficient for the NACA 0018 airfoil, Experimental Data from [7].

Figure 4 displays the pressure contour plot for coarse meshing and fine meshing. The contours in these figures look very similar but as explained earlier, the inaccuracies are in the boundary layer region in the coarser mesh. Moreover, the $k - \varepsilon$ model which was used is based on wall-functions and is not sufficiently accurate when the flow is separated. This occurs earlier in the high angle of attack cases justifying the increased error [8].
The CFD analysis is utilised primarily as a means of determining which design gives the best performance qualitatively. Due to the limited computational resources available, a mesh with comparable characteristics (such as $y^+$) to the coarse mesh case was used in the analysis for the design of the FSAE MUR 2009. The 3D CFD model made use of symmetry along the centreline of the car, thereby halving the computational domain. The boundaries are set to be approximately 4 to 5 car lengths in order to avoid any undesirable effects from the boundaries as shown in Fig. 5. The ‘wall ground’ is set to have a boundary condition as a moving wall in the FLUENT environment to simulate the motion of the car. The meshing is generated on Gambit with about 1,500,000 elements on each numerical model using triangular and tetrahedral cells.

Fig. 4. Pressure Contours Plot at $\alpha = 20^\circ$.

3. Improving Airflow

3.1. Varying the sidepod inlet

To investigate the effect of the inlet size on the heat transfer rate, $Q$, the inlet size is varied whilst the outlet size is maintained, reducing the number of variables within the analysis. Since airflow behind the wheel is turbulent, one may expect the airflow to be improved for smaller inlet sizes. By reducing the inlet size however, the mass flow rate may drop due to the reduced area for airflow.

Fig. 5. Boundary Conditions Used in FLUENT.
In order to obtain a greater $Q$, it is crucial that inlet area be analysed in order to draw insight into the relationship between airflow flow rate and inlet size. Figure 6 displays streamlines released into the sidepod inlet for different inlet sizes. In case A, the width of the inlet size is the same as the opening between the nosecone and the front wheel. In case B, the inlet is made smaller to avoid the turbulent airflow behind the wheel, while case C, the inlet is made wider. Turbulent airflow is evident at the inlet of the sidepod in case C, displayed in Fig. 5. Conversely, the airflow in case A and B is more ordered and less chaotic compared to case C.

(a) Case A: Sidepod Inlet Size the Same as the Gap between the Wheel and the Nosecone.

(b) Case B: Small Inlet Size.

(c) Case C: Big Sidepod Inlet Size.

Fig. 6. Streamlines (Coloured by Velocity) for Varying the Sidepod Inlet Size.
As discussed previously, heat transfer rate is also affected by the area of the sidepod inlet as it determines whether the flow into the sidepod is laminar or turbulent. Table 2 presents the value of $Q$, lift and drag forces for each design. It is apparent that values of $Q$ for case B and C are superior when compared with case A. These results suggest that the mass flow rate through the radiator cannot be optimised by increasing or decreasing the inlet size. Other variables such as the quality of the airflow have to be considered. In case B, which has a smaller inlet size, a higher flow rate is observed due to the superior quality of the airflow into the sidepod.

Table 2. Heat Transfer Rate, Drag and Downforce for Different Inlet Sizes.

<table>
<thead>
<tr>
<th>Case</th>
<th>Mass Flow Rate $m$ (kg/s)</th>
<th>Heat Transfer Rate from Radiator $Q$ (W)</th>
<th>Drag $F_D$ (N)</th>
<th>Downforce $-F_L$ (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>0.4231</td>
<td>9776</td>
<td>33.0</td>
<td>3.5</td>
</tr>
<tr>
<td>B</td>
<td>0.4329</td>
<td>9983</td>
<td>32.3</td>
<td>5.7</td>
</tr>
<tr>
<td>C</td>
<td>0.4246</td>
<td>9866</td>
<td>36.7</td>
<td>-9.0</td>
</tr>
</tbody>
</table>

Notably however, the results suggest that although the value of $Q$ increases, it only increases marginally by about 2%. On the contrary, drag force associated with different cases deviates substantially. Case C shows an increase in net drag force of 11% and 14% compared with cases A and B respectively. For case C, the body is less streamlined and the flow near the sidepod inlet is increasingly turbulent. Downforce accompanied with case C is the least of the three cases. Reasons for the large differences are due to the close proximity of the front wheel and sidepod inlet which induces a low pressure region over the top surface of the car thereby inducing a net lift force.

3.2. Varying the sidepod inlet

Changing the inlet size will increase the heat transfer rate $Q$, but the inlet area is insufficient to aid heat management. To further investigate the effect of larger inlet size on heat transfer rate $Q$, the front half of the sidepod is removed (Case D), thus creating an appreciably shorter sidepod. This was performed to maximise the amount of air entering the inlet. For case E, the undertray is removed to see the effect of the front undertray on the cooling performance. Reverse streamlines are shown in Fig. 7, for both case D and case E. There exists negligible difference in airflow between case D and E where both instances present minimal turbulence. An increase in the inlet area, however, increases heat transfer rate $Q$ by approximately 20% for both case D and case E compared to case A as is evident in the results shown on Table 3.

Table 3. Heat Transfer Rate, Drag and Downforce for Short Sidepod Designs.

<table>
<thead>
<tr>
<th>Case</th>
<th>$m$ (kg/s)</th>
<th>$Q$ (W)</th>
<th>$F_D$ (N)</th>
<th>$-F_L$ (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>0.4231</td>
<td>9776</td>
<td>33.0</td>
<td>3.5</td>
</tr>
<tr>
<td>D (with undertray)</td>
<td>0.4958</td>
<td>11513</td>
<td>36.2</td>
<td>7.8</td>
</tr>
<tr>
<td>E (no undertray)</td>
<td>0.4847</td>
<td>11318</td>
<td>39.7</td>
<td>-22.1</td>
</tr>
</tbody>
</table>
Fig. 7. Streamlines (Coloured by Velocity) and Contours of Pressure for Short Sidepod with and without the Front Undertray.

(a) Case D: Short Sidepod with an Undertray.

(b) Case D: Short Sidepod with an Undertray.

(c) Case E: Short Sidepod without an Undertray.

(d) Case E: Short Sidepod without an Undertray.
In the pressure contour plots in Fig. 7, one may see that for case D there is a tendency toward high pressure regions on the top surface of the undertray. This, in effect pushes the undertray and thereby the car down. As a result, the downforce generated increases by more than 100% compared to case A. Without the front undertray however, the high pressure region on the top half of the car is eliminated. The vehicle, as a result, produces notably higher lift in case E, which is undesirable. Moreover, the drag generated is also high in case E with an increase of about 20% which can be attributed to the removal of the front undertray.

In short, it is certainly superior to develop a short sidepod and keep the inlet reasonably large, possibly with a front undertray feature to gain downforce. The larger inlet size will compensate for the fact the car usually runs at lower speeds. The front undertray will then assist with cooling by preventing air from escaping under the car in front of the radiator, and will also keep drag values low and induce substantial downforce.

### 3.3. Gills and chimneys

The 2009 MUR vehicle has the radiator mounted inside the sidepod at an angle of attack of 45° for two reasons; firstly to increase the size of the radiator core for greater heat transfer, $Q$, and secondly to take advantage of the natural convection of air [2]. Once the air exits the radiator, it travels directly upward toward the top surface of the sidepod. This will restrict the airflow, creating low velocity regions inside the sidepod as shown in Fig. 8.

![Fig. 8. Contours of Velocity Magnitude inside the Sidepod.](image)

In order to solve this problem, gills should be included in the region where the sidepod is blocking the airflow (Case B2). Gills are designed to assist the expulsion of hot air from the radiator exit, allowing the sidepod to breathe. Similar to gills, chimneys are analysed in case B3 as a means to aid the release of air from the sidepods. Figure 9 displays the velocity contour plots for Cases B2 and B3. From these plots, one may view the effects of gills and the considerable improvements to airflow inside the sidepod. The low velocity region inside the sidepod is appreciably reduced. Additional chimneys, as in case B3, further improve the airflow by marginally increasing velocity. As a result, $Q$ in Case B2 and B3 increases by 33% and 46% respectively, as portrayed in Table 4.
Table 4. Heat Transfer Rate, Drag and Downforce for Addition of Gills and Chimneys.

<table>
<thead>
<tr>
<th>Case</th>
<th>( \dot{m} ) (kg/s)</th>
<th>( Q ) (W)</th>
<th>( F_D ) (N)</th>
<th>(-F_L) (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>B</td>
<td>0.4329</td>
<td>9983</td>
<td>32.34</td>
<td>5.72</td>
</tr>
<tr>
<td>B2 (gills)</td>
<td>0.5961</td>
<td>13303</td>
<td>34.12</td>
<td>19.12</td>
</tr>
<tr>
<td>B3 (gills with chimney)</td>
<td>0.6583</td>
<td>14567</td>
<td>35.7</td>
<td>22.1</td>
</tr>
</tbody>
</table>

Fig. 9. Contours of Velocity after Adding Gills and Chimney to the Sidepod.

Gill and chimney features have proven to be very effective in increasing heat transfer rate \( Q \) with a very low drag penalty. Drag in case B2 and B3 is only marginally higher from case B, 5.5% and 10.4% respectively. Interestingly however, the added downforce is a result of the addition of gill and chimney features. Downforce generation increased considerably for both cases as compared to case B. This can be partially attributed due to the negative thrust generated by the flow through the gills and chimneys similar to that of a aircraft jet engine.

3.4. Suspension covers

The 2008 MUR vehicle features a suspension cover placed underneath the monocoque to deflect airflow around the suspension components. Although it is very effective in protecting the suspension parts, the cover actually adversely affects the airflow underneath the car as depicted in Fig. 10. Airflow towards the suspension cover is forced around the side of the car. As a consequence, the airflow interferes with the upstream flow entering the sidepod inlet. Such interferences create turbulent flow around the bottom half of sidepod inlet (Fig. 11). This then causes the heat transfer rate, \( Q \), to reduce.
In order to reduce the effects of this problem, the bottom of the nosecone is made flat and the suspension cover has been removed. By doing so the airflow has improved drastically as displayed in Fig. 11. Less turbulence exists at the bottom of the sidepod inlet. Consequently, \( Q \) has increased by about 12% by improving the airflow without making any major changes to the sidepod, as shown in Table 5. Accordingly, for maximum cooling performance, it is important that there is no turbulent airflow generated upstream of the sidepod.

**Table 5. \( m \) and \( Q \) for Different Airflow Conditions.**

<table>
<thead>
<tr>
<th>Case</th>
<th>( m ) (kg/s)</th>
<th>( Q ) (W)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bad Airflow (Case C)</td>
<td>0.4246</td>
<td>9866</td>
</tr>
<tr>
<td>Good Airflow</td>
<td>0.4847</td>
<td>11029</td>
</tr>
</tbody>
</table>

---

**Fig. 10. Suspension Cover in MUR 2008.**

**Fig. 11. Streamlines (Coloured by Velocity) Plot for Case B2 and B3.**

(a) Bad Airflow (with Suspension Cover).

(b) Good Airflow (without Suspension Cover).
3.5. Ducting air to the engine

Since 2006, MUR has utilised a Honda CBR 600RR motorbike engine as its power plant. As it is a motorbike engine, it is designed to operate in an open environment, this is why fins are located at the surface of the engine to assist with cooling. The full monocoque design in 2008 had an enclosed rear and most of the engine was covered by the chassis. As a result, very little airflow is supplied to the surface of the engine generating a non-ideal operating environment. The MUR 2009 sidepod, therefore, is designed to direct some of the air to flow to the engine to further assist in cooling as depicted in Fig. 12. From Fig. 12(b) which gives a better representation of the streamlines with the sidepods and nosecone (body kit) hidden, it can be seen that the rear end of the sidepods direct the airflow inwards towards the engine.

![Diagram showing ducting air to the engine](image)

(a) Ducting Air to the Engine.

(b) Body Kit Hidden.

Fig. 12. Reverse Streamlines Coloured by Temperature Released from the Engine.

4. Conclusions

A summary of $Q$, drag and downforce for different designs is shown in Figs. 13 and 14. It can be concluded that for maximum $Q$, the sidepod inlet area should be reasonably large to ensure maximum airflow while considering factors such as the turbulence effect from the wheel. Although a larger inlet causes an increase in drag, adequately cooling the engine is more important than reducing drag. The
addition of gills and a chimney improves cooling with a very low drag penalty. An increase in $Q$ of 33% and 46% were observed for the additions of gills and chimneys respectively. In addition it is very important to keep the flow as laminar as possible by removing the parts in the car that will induce turbulent airflow, this especially includes attachments underneath the car such as suspension covers.

![Heat Transfer Rate $Q$.](image1)

**Fig. 13. Heat Transfer Rate $Q$.**

![Drag and Downforce.](image2)

**Fig. 14. Drag and Downforce.**

References


