

Race Car Aerodynamics:

The Effect of Cooling System Layout on Race Car Performance

### Project by Kellieann Hukins

#### 0625255

# Progress Review

#### *Project in Association with Perry Cohn of Angle Consulting.*



Table of Contents

[1.0 Introduction 1](#_Toc219265062)

[2.0 Project Outline 2](#_Toc219265063)

[3.0 Plan 3](#_Toc219265064)

[3.1 CFD Simulation 3](#_Toc219265065)

[4.0 Current Progress 4](#_Toc219265066)

[5.0 Further Progress 5](#_Toc219265067)

[6.0 Intentions 7](#_Toc219265068)

[7.0 Project Timetable 8](#_Toc219265069)

# Introduction

The project will assess the effects of airflow through a race car cooling system, and how this affects the performance of the car in a similar way as the airflow over a car also affects the performance. Performance of the car is also affected by mechanical effects of the cooling system layout; fluid pressure drop through the radiator and oil cooler (the lower the pressure drop, the less the power lost in pumping the fluid through the cooler and radiator), and moments of inertia, including centre of gravity and polar moments, are two key areas of interest.

To assess the performance, focus will be on engine power, weight and drag. The greater the operating temperature of the engine, the lower the power output. However, the greater the operating temperature of the engine, the smaller the cooling system is; which means less weight and less drag. Reducing drag will increase car performance. To provide maximum car performance a balance between the three factors (drag, operating temperature and engine power) will be required.

This is particularly important with the standardized engine rule which is about to be introduced into Formula 1 racing where engine life is critical; a balance is essential as to not run the engine too hot which reduces power and life, but decreasing cooling system drag is desirable.

# Project Outline

An engine has a power output and also a heat rejection. The heat rejection of an engine is usually measured at 2 or 3 temperatures; for example 80, 100 and 120 degrees C. For each of the operating temperatures, the power output of the engine will be different, usually producing greater power at the lower operating temperature.

However, the higher the temperature the engine runs at, the smaller the heat rejection, and hence the smaller the radiators required. This then leads to a reduction in aerodynamic drag, which means that the car can go faster despite having less power.

The balance is therefore finding the optimum cooler/radiator size without causing the engine to overheat.

Clearly, the core needs to be able to dissipate the same, or more heat, than that produced by the engine as heat rejection. If the cores are too small, the engine will overheat.

The aim of this project is to investigate the balance between operating temperature, power and drag in order to maximise performance.

# Plan

Run CFD software to simulate airflow through the cooling system in order to analyse the cooling system layout. Reducing the radiator and oil cooler core sizes would reduce the weight and drag of the cooling system. However, it is essential that the core is still capable to dissipate the required heat.

Analysis of data will be conducted using traditional methods, including graphs and charts.

Aerodynamic analysis of the cooling system geometry will be conducted with fluid flow software, ***Star CCM+***, using geometry in a *Para-solid* format through ***Solidworks***software.

It is required that the geometry will be analysed and edited accordingly and thus re-run in Star CCM+ to investigate the change in layout.

Analysis of all data and results will be used to aid research and simulation to attempt to evaluate the given geometry to improve car performance.

# CFD Simulation

Theoretically, if the engine runs at a set temperature (rejects a set heat) then the cooling system is required to transfer that set amount of heat. To transfer that set heat the cooling system must be of a given area. This can be calculated.

Realistically, the duct shape and the position of radiator will affect the flow of the theoretical fluid and the cooling system may not receive the uniform flow over the area of the radiator core. The radiator may not actually transfer enough heat. The best way to ensure efficiency is to ensure that the entire core is cooling the same amount (uniform flow).

Without CFD simulation or wind tunnel testing, fluid flow distribution across the surface of the core, and thus how much of the core is actually being useful, can only be assumed.

# Current Progress

Some background research has been conducted using resources such as textbooks and reliable Internet websites. Main sources detailed in Logbook. Exploration of possible journals and papers has been unsuccessful; Logbook gives details of searches and results. Some useful papers have been found however it does not seem possible to obtain these without an incur in cost.

Geometry of standard cooling system has been obtained. The geometry is general and thus neither specific to F1 or A1.

Typical data including real cooling system data and estimated engine data has been collected and received. A Spreadsheet has been compiled for the calculation of heat rejection, based on the equation;



Where;

*Q* - Heat transferred (J)

*m* - mass (kg)

 - specific heat capacity (at a constant pressure) (J kg-1 K-1)

 - Change in temperature (K)

Calculation is dependant on core areas and fluid velocity through the duct.

Tutorials for Star CCM+ simulations have been completed and revisited in order to learn and familiarize the software. Introductory tutorials have been attempted including heat transfer tutorials and setting up & running a simulation. Tutorials have been problematic & thus so far have not resulted in a successful simulation of the cooling system geometry. The key problematic area seems to be with importing and meshing geometry. Trial & error with import settings and file formats has not been beneficial. Full details are logged throughout the Logbook.

# Further Progress

Ideally, the speed of air in the duct should be between 7 and 10 m/s, regardless of car speed. The faster the air flow through the core, up to 10m/s, the more heat will be rejected by the cooling system; it is the speed of the airflow through the radiator core, and the flow distribution, that determines the amount of cooling. Flow of greater speeds than 10m/s will increase the drag of the system. The pressure distribution will show the flow across the core surface. A uniform pressure distribution will ensure maximum cooling (for any velocity).

To achieve the required air speed through the duct, two simulations will be necessary:

1. Flow through a domain with the duct inside to test flow speed through the duct (mass flow rate). If the flow rate is not within the required limits the duct will need to be edited to achieve this.

(May need to model a solid volume and ‘extrude cut’ the duct from the volume. This would simulate the fluid only).

1. Flow through the duct at given air flow speed. Once the air flow speed is suitable, the flow can be simulated through the duct with the parameters set to give the relevant inlet velocity.

(The current geometry may need to be altered from a surface to a solid in order to mesh the geometry).

To test the air flow distribution across the surface of the core, the pressure distribution should be modelled. Using data from the spreadsheet will show the required cooling and heat transfer; the simulation will not show this as typical data can be given from the spreadsheet. The simulation will only show that the two cores receive the required flow to provide the necessary cooling, and thus that the air flow distribution is not assumed. If the airflow distribution is not uniform the duct shape will need to be edited in and simulated until a uniform (or closest to a uniform) airflow is achieved.

With a uniform airflow the relevant geometry can be assumed to have the optimum cooling for the given core areas. However this may not be the optimum balance between cooling system drag and weight. The system can be re-analysed with the data in order to see if the layout can be edited to improve car performance (reduce weight, drag but still maintain enough flow as to transfer required heat from engine at original set temperature). The data should be presented in a 3D graph in order to best analyse this.

# Intentions

**How will I run the simulation?**

The geometry represents the radiator and oil cooler as blocks. These can be treated as porous media by the CFD software, and they are characterised by an air pressure drop through the core as a function of flow velocity. The speed of the air through the core, as well as the size of the core, will determine the amount of heat that can be dissipated by the core.

**What results am I trying to collect from my simulation?**

What happens to drag as the core size reduces? Will the reduction in drag be offset by the reduction in engine power, and will the car go faster?

**How will I get the results?**

CFD results will tell me what airflow I am getting through the core, as well as flow distribution.

Based on flow velocity, and core data the heat rejection can be calculated (using the spreadsheet).

**What will the results tell me?**

This will tell me if the car will go faster or slower.

**What will I do with the results?**

Collate the data, into a 3D graph, with axes of drag, core area and operating temperature, based on a given car speed. (To find the optimum balance: Greater power at the lower operating temperature, less drag (reduced area/weight) at higher operating temperature).

**What will I be varying to get the required results?**

Air flow velocity and core areas.

# Project Timetable

In order to reach the deadline for Project draft submission (Week 22), I intend to make progress on my draft report throughout this term alongside the main body of the Project work (CFD simulations & Data analysis). However, in order to complete a significant amount of work before the draft report is due in, the simulations will need to be completed as soon as possible. Completion of the first simulation is now overdue with the initial timetable target of Week 10; this set back has been caused by problems in setting up and running the cooling system geometry in a Star CCM+ simulation. Fortunately this current set back should not affect my overall progress as long as the simulations are run in the near future. In an attempt to achieve this I will be running the new simulation tutorial.