Henri Siitonen

IMPROVING THE AERODYNAMICS OF A COOLING SYSTEM OF A FORMULA STUDENT CAR

Bachelor's Thesis
Automotive- and Transportation Degree

December 2011

MIKKELIN AMMATTIKORKEAKOULU
Mikkeli University of Applied Sciences

DESCRIPTION

MIKKELIN AMMATTIKORKEAKOULU Mikkeli University of Applied Sciences	**Date of the bachelor's thesis** 12/13/2011
Author(s) Henri Siitonen	**Degree program and option** Automotive- and Transportation Degree

Name of the bachelor's thesis

Improving the aerodynamics of a cooling system of a Formula Student car

Abstract

Now days the aerodynamics is the last frontier in which the race teams try to gain those precious seconds and levels of grip. The rules try to make the playing field even so that the competition would be fair. Still the teams come up with new innovations which then will give them the edge. For an automotive engineer it is vital to know the basics in aerodynamics, because also in the commercial side the cars are made more aerodynamic in order to achieve the lowest possible fuel consumption.

In SAE Formula Student the aerodynamics haven't played a big role in the past, but as the rules are very strict regarding the engine, powertrain, etc. what is left is the aerodynamics. Only few teams have incorporated aerodynamics in their design and some of those teams have been quite successful. The main reason why the teams haven't spent time in developing the aerodynamics is the low speed in which the car operates. But this doesn't mean that the aerodynamics should be forgotten, it just means that one shouldn't focus in getting more downforce but to make the car more streamline to reduce the drag. Also use it to improve the cooling system!

In this thesis the basics of aerodynamics will be presented briefly, also the basics of an internal combustion engine cooling system. Then the different ways of measuring the air flow and air speed is explained. These different ways of measuring are then evaluated, in according to which of them are suited to be used in the tests. Lastly the actual modifications that were done to the car and the results that were achieved.

Subject headings, (keywords)

Cooling, System, Formula, Student, Aerodynamics

Pages	**Language**	**URN**
44	English	

Remarks, notes on appendices

Tutor	**Bachelor's thesis assigned by**
Juhani Martikainen	

CONTENTS

1 INTRODUCTION

Aerodynamics plays a big role in today's motorsport. The teams and the manufactures spend countless hours in the wind tunnels to make their car produce the least amount of drag but at the same time getting the most downforce that they can. The aerodynamics is the last frontier where improvements can be made and getting the right settings for the right track, could spell win or lose.

In the Formula Student the regulations are relatively strict regarding engine size, drive train, etc. One neglected field is the aerodynamics and this is not regulated much. So teams can be as innovate as they want, but not many teams utilize aerodynamics and take it into consideration while they are designing the car. When the engine displacement is limited and the maximum dimension for the car is given, the limits are reached sooner or later. Then focusing on the aerodynamics is a must. This doesn’t mean that the goal should only be to get more downforce, it could also be trying to get the car more streamlined, thus reducing the drag of the vehicle.

This paper focuses on the improvements that can be made in a Formula Student car and the measurements to decide which modifications actually work. The main points are to improve the actual cooling power, the air flow that exists in the cooling duct and the drag and pressure losses in the cooling duct. These points will be discussed in this thesis.

The first part of the thesis focuses on describing the basics of the cooling systems and the different measuring methods that can be used in measuring the air flow. The second part describes the modifications that were made to the test car, and why these particular modifications were chosen. Last part then sums up the results from the measurements and of the conclusions that can be made from this study.

1.1 Cooling systems

The internal combustion engine used today is far from being the most efficient, efficiency being between 25-35%. Out of 1 liter of fuel only 3dl is used to actually move the car, rest of the fuel goes to waste. There are different losses in the engine but the biggest loss comes in terms of heat. Roughly 60% is used as heat. From this 60% little more than half is exhaust gases, but the rest is left for the cooling system.

In an ordinary passenger car that produces around 100 kW, 30 kW is used to move the car, 35 kW is put out though the exhaust and 28 kW is left for the cooling system to get rid of. 28 kW is a lot of heat to handle, so the cooling system has a hard task. For the engine itself it's vital to have steady operating temperature. To get the best efficiency out of the engine, the engine should be working between 85-95 °C. When temperatures go above that, the wear of the engine increases dramatically, parts start to expand more, etc. And if the temperature goes high enough the engine will cease up.

In an ordinary passenger car the cooling system consists of 11 parts. These parts are portrayed in the following picture.

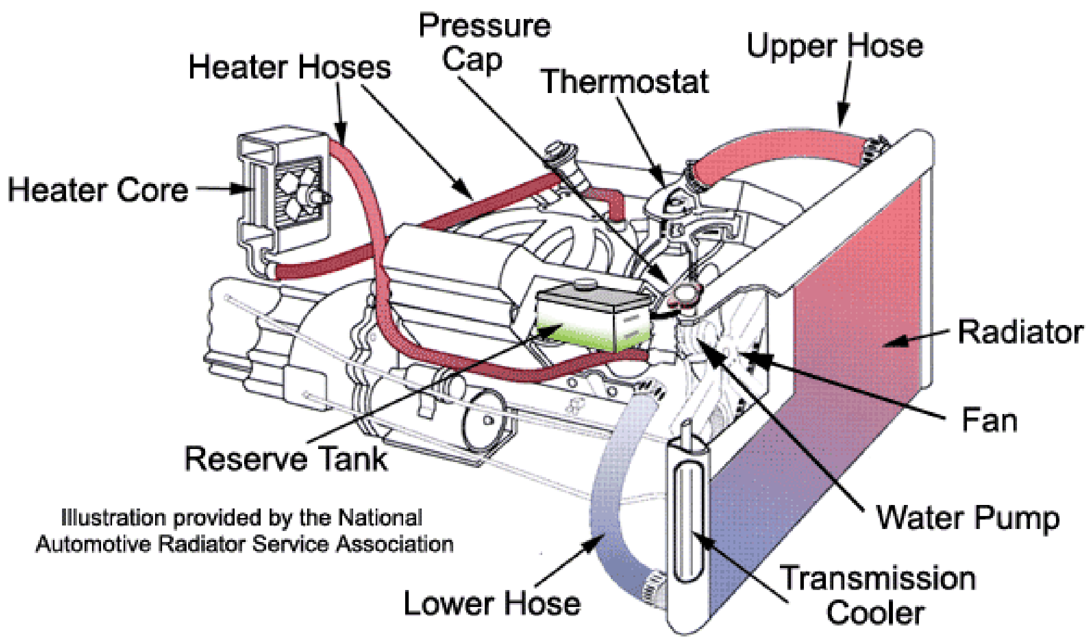


Figure 1 Typical engine cooling system /01/

The cooling system looks pretty simple. There are not too many parts and the functions of all the parts are understandable. The radiator is the part where the water is cooled. All of the radiators are air-cooled, that means that the air going through the radiator takes the heat away from the water. So the air flow through the radiator is really important. The main source of air flow is formed by the movement of the car, but when the car is standing still the fan makes up the air flow.

In a racing car the cooling system is a little bit more complicated. In an ordinary passenger car the weight, the space, the cost, etc. are important too, but in a racing car these things are even more highlighted. The biggest problem by far is the space. In an open-wheel race-car the space tends to be really tight, but it is still required to fit the large enough radiator in order to have required cooling power. Then the next problem is the coolant pump. To get the most out of the engine with the least amount of waste power, the pump needs to be just powerful enough to have the required coolant flow. So oversized coolant pump is not preferable.

2 PROJECT DESCRIPTION

The project car was from the Hochschule Esslingen race team, Rennstall. The name of the car is Stallardo and it was the 2010 model. The car had a Honda 4-cylinder engine, with a displacement of 599 cc^3. The cooling was taken care by two radiators, one main radiator and one smaller to help the cooling. The oil cooling was also done with the same radiators. The cooling system had one electronic water pump to circulate the water. The frame consisted of tubes and carbon-fiber panels were used to cover the tube-frame.

In the car the cooling was a major problem. The radiator that was designed for the car was too small. After the first race it was obvious that the cooling system needed to be improved. The operating temperature for an internal combustion engine has to be between 85-95 °C. On the ’10 Stallardo the coolant temperatures were constantly between 100-110 °C, peaking even at 120 °C. /02/ The team added another smaller radiator under the main radiator to help with the cooling. Even adding this extra radiator the cooling power just wasn’t enough. The car was still overheating, so something had to be done.

Two teams were given the task to improve the cooling. One team focused on the actual cooling system and one team had the task of improving the aerodynamics and gathering important information about the aerodynamics. Our team consisted of me and two other persons. This project was done for the team in the summer of 2011.

Aim of the project was to improve the cooling system by improving the air flow to the radiator. Any changes to the original cooling system were out of the question. The only thing that could be done was to modify the existing cooling duct with bolt-on parts, that didn't leave any permanent marks. By maximizing the air flow and pressure rise the cooling capacity of the existing radiator could be improved.

Another aim was to collect information about the aerodynamics of a Formula Student car and do vital base testing for the future. This information then can be later used in additional tests. The test was done by carrying out a static test on the car, building the necessary components to modify the cooling duct and the wind tunnel used in the test. Also gathering information from different sources and using them to validate the results. In the next chapter the different measuring methods are explained and evaluated how can they be utilized in the tests.

3 MEASURING METHODS

In order to assess the cooling system and to see if any improvements were achieved, it’s crucial to do some measurements. Cooling performance wise, two things matter the most. Firstly the air side, so the actual air flow through the radiator and then the mass flow of liquid inside the cooling system. In this thesis the focus is on the air side so the ways of measuring the air flow and the air mass flow will be presented.

There are different ways of measuring the air flow and the air mass flow. In the following chapters the most useful ones are listed. There are also 2 performance parameters that are useful in evaluating the cooling system performance.

There are a few problems that make the measuring more complicated in real-life situations:

- The air flow is not laminar
- The different heat sources affect the results
- The conditions are not stable

3.1 Differential static- /total-pressure measurements

Pressure measurements can be made mainly with three different probes. These probes are Static-, Prandtl- and Kiel-Probe. All of these probes are suitable to be used in the wind tunnel tests and also in the real-world tests.

This pressure measurement requires the measurement of two different pressures, static- and dynamic pressure. This can be achieved with just one probe (Prandtl-tube or Kiel-probe) or with two different probes (Static and Dynamic). The physics behind this is that measuring the total pressure in front of the probe and then measuring the static-pressure from the free-stream further below and then calculating the difference between these two, results in dynamic pressure. From the dynamic pressure it's possible to calculate the air speed using the following equation:

$$v = \sqrt{\frac{2}{\rho}\Delta p_{\text{dyn}}}$$

Where:

v = the air flow velocity

ρ = the air density

Δp_{dyn} = the dynamic pressure

The biggest drawback with the Static/Dynamic or the Prandtl probes is that they are highly sensitive to the changes in the angle in which the air flow meets the probe. The angle has to be kept below 12° (comparing it to the horizontal level). Below this limit the error introduced is roughly ~1%. Also unsteady flows affect the accuracy of these probes. One option is to use the Kiel-probe that is relatively insensitive to these conditions. In a Kiel-probe the head is placed in a Venturi-tube to achieve laminar flow to the head. With this probe the yawing angle can be up to 60°.

In summary these measurements are usable in static and road tests, but the acquisition of these probes can be difficult and very expensive. Also the mounting of these probes imposes additional problems, especially if the original design can't be modified. So these things have to be taken in to consideration when designing the test program.

3.2 Thermal based measurements

Thermal based measurement basically consists only of the Hot-Wire-Anemometer (HWA) or sometimes referred as the Mass-Air-Flow meter. The Mass-Air-Flow meter is more widely known in the automotive industry. The HWA consists of a thin wire that is positioned in the air flow and a small current is put through the wire. The wire heats up and the air flow around the wire cools it down. The cooling power is dependent on the speed of the air flow. By measuring the resistance in the wire it's possible to determine the mass flow rate. Usually these kinds of sensors are small, they have rapid response to changes in the speed and even at slow speeds they are accurate.

But this kind of measurement method has a few major flaws that prevent its use in on-the-road testing and in wind tunnels:

> "- The hot wire itself is very fragile. Dust, dirt, or gravel pieces can easily destroy the hot wire.
>
> - Using a HWA without knowing the correlation of airspeed, temperature and electric resistance would cause a complex set-up calibration.
>
> - Due to the fact that the measuring method is a thermally based system it needs a stable environment under stable conditions. Placing the HWA in front of or behind the radiator implies that several heat sources influence the measurement.
>
> - Using a HWA requires a laminar airflow in the cooling duct for reliable results. Turbulent airflow would falsify the test results.
>
> - High costs" /03. p.18/

3.3 Vane anemometer

The vane anemometer consists of a small vane wheel that is positioned inside a housing, which may have different shapes. The incoming air flow rotates the wheel and the speed of the wheel is then measured. It will directly indicate the speed of the air flow.

The vane anemometers can be used to measure the speed of the incoming air and from that it can be calculated how much mass flow there actually exists, when the density of the air and the area of the inlet are known. In order to achieve reliable results the vane anemometers have to be small enough and there must be plenty of them to cover the whole radiator. This way the whole flow area is measured.

The biggest problem with the vane anemometers is that if the flow is not uniform or turbulent, the accuracy of the measurements will be reduced. The turbulent flow changes the speed of the vane depending on which angle the turbulent flow hits the blades. Also if the cooler has a fan behind the radiator it's impossible to install this array of anemometers, because of the space requirements.

Figure 1 A rays of vane anemometers /04/

3.4 Performance parameters

The next two ways of assessing the cooling power rely completely in math and are not directly measured. Both of these only require temperature measurements, so they are ideal for on-the-road and static tests. Also it makes them really cheap to use, so they are perfect for student-teams.

3.4.1 Air-To-Boil (ATB)

"The Air-To-Boil temperature is defined as the ambient temperature at which the coolant temperature at the radiator inlet reaches the boiling point. In fact ATB provides a measure of how far the cooling system is from boiling." /04, p.4/

ATB is defined as:

$$ATB = T_{BP} - (T_{CI} - T_{AI})$$

Where:

T_{BP} = coolant boiling point

T_{CI} = coolant radiator inlet temperature

T_{AI} = air radiator inlet temperature

"The ATB-parameter can only be measured / calculated during vehicle tests when the whole cooling system has achieved a stable operating point. It is a costly and time-consuming process, since it requires stable ambient and engine load condition." /04, p.4/ ATB does not directly give the cooling airflow rate but it can indicate the effect of changes in the airflow on the cooling performance.

3.4.2 The Specific Dissipation (SD)

"In contrast to the ATB, the Specific Dissipation (SD) is relatively insensitive to changes in the ambient and coolant temperatures." /03, p.31/

$$SD = \varepsilon * C_{min} = \frac{Q_c}{Q_{max}} * C_{min}$$

Where:

ε = heat exchanger effectiveness

C_{min} = minimum capacity rate

$$Q_c = m_c * c_{(p,c)} * (T_{(ci)} - T_{(co)})$$

$$Q_{max} = m_a * c_{(p,a)} * (T_{(ci)} - T_{(ai)})$$

$$C_{min} = m_a * c_{(p,a)}$$

Where:

T_{ci} = radiator coolant inlet temperature

T_{co} = radiator coolant outlet temperature

T_{ai} = radiator air inlet temperature

T_{ao} = radiator air outlet temperature

$c_{p,c}$ = coolant specific heat

$c_{p,a}$ = air specific heat

"SD is defined as the heat transfer rate of a heat exchanger divided through the maximum temperature difference across the heat exchanger." /04, p.4/

$$SD = [m_c * c_{(p,c)} * (T_{(ci)} - T_{(co)})]/(T_{(ci)} - T_{(ai)})$$

Ignoring losses or thermal radiation the mass airflow can be approximated by stating:

$$m_c * c_{(p,c)} * \left(T_{(ci)} - T_{(co)}\right) = m_a * c_{(p,a)} * (T_{(ao)} - T_{(ai)})$$

$$m_a = \left[m_c * c_{(p,c)} * \left(T_{(ci)} - T_{(co)}\right)\right]/[c_{(p,a)} * \left(T_{(ao)} - T_{(ai)}\right)]$$

The disadvantage of using performance parameters is that the airflow is not measured directly but calculated. So there is always a bigger error included in the results. However this approach has also several advantages:

- Both the financial and the time effort for the test setup are fractional. ATB requires two temperature sensors, SD four

- Both are suitable for on-the-road tests

- SD parameter is much more insensitive to the changes of the environment

3.5 Summary of the measuring methods

Chart 1 compares the different methods for measuring the airflow rate through a vehicles radiator presented on the previous pages. The comparison and the assessing of these different methods are rated depending on four attributes. These are cost, time-effectiveness, accuracy and lastly how suitable they are for on-the-road testing. The CFD simulation wasn't done in this project, but it still left there in the chart so that one can see the pros and cons of CFD.

	Costs	Time-effectiveness	Accuracy	Suitable for on-road tests
Pressure measurements				
Differential static measurements (pressure drop)	-	-	-	+
Differential static and total pressure measurement (Kiel probe + Pitot static tube)	--	--	-	+
Thermally-based systems				
Hot-wire anemometers	++	•	+	--
MAF sensors	-	-	-	+
Vane anemometry				
Vane anemometers	++	+	+	--
Array of vanes applied on the radiator	-	-	•	+
CFD	++	--		
Performance parameters				
ATB	++	+	-	++
SD	++	+	+	++

Chart 1 The different measuring methods (CFD wasn't included)

The SD-parameter is an indicator of the airflow rate through a radiator. If the SD-value of a reference configuration is measured and subsequently a change is made, then the SD-parameter shows a resultant cooling effect caused by the different airflow rate. It is suitable both for static and on-the-road tests. The test results from static tests are therefore comparable with the test results from on-the-road tests when the car is operating under real-life conditions. Compared to the other measuring methods the financial and time efforts are minimal. In the next chapters the static test, the test bench and the wind tunnel and the aerodynamic modifications will be explained. Main focus is on the aerodynamic modifications, so the test bench and the wind tunnel will not be covered in too detailed fashion.

4 STATIC TEST

4.1 Aim and accomplishment

The aim was to test the different modifications in a real-life situation and study the air flow inside and outside the cooling duct. This was done by having the car in front of the wind tunnel and blowing air towards the cooling duct, while the engine was kept at a constant rpm to heat the water inside the cooling system. All of the modifications were tested with and without the fan and the wind speed at the inlet was measured.

4.2 Test bench and the flow straightener

The test was done in the automotive-faculty test laboratory in the Hochschule Essling-en. In the laboratory was a fan what was suitable for the test. Suitable meaning in this that it could provide enough wind speed so that the result would be useful. The aim was to achieve roughly 15 m/s wind speed, because that is roughly the average speed that the car has while driving the track. The fan was able to produce wind speeds be-tween 20-25 m/s, but that was without the flow straightener. The only problem was that the flow was highly turbulent and the speeds weren't uniform. The solution was to build a flow straightener.

Figure 2 The flow straightener and the test bench

The flow straightener takes the spin out of the flow, 'thus introducing a uniform flow to the car. This is crucial in getting trustworthy results. If the air flow is not laminar before it actually gets to the car the results would be affected greatly, because when the air is turbulent its attributes are completely different. The Reynolds-number is much higher and also the drag induced falsifies the results. In order to keep the performance losses to the minimum the pressure loss in the duct has to be as small as possible. This was obtained by not having sharp corners or any other sharp surfaces that would introduce drag to the system.

With the flow straightener the wind speeds dropped quite significantly. The average wind speed was only 10 m/s, and this was much less than desired. In the following chart 2 the different wind speeds can be seen. The position, in the left side of the chart, means the distance from the centre of the fan to either side of the fan. The blue line shows the air speed without the straightener. The green and pink line shows the air speed with the straightener, measured so that the green line represents the vertical and the pink one horizontal.

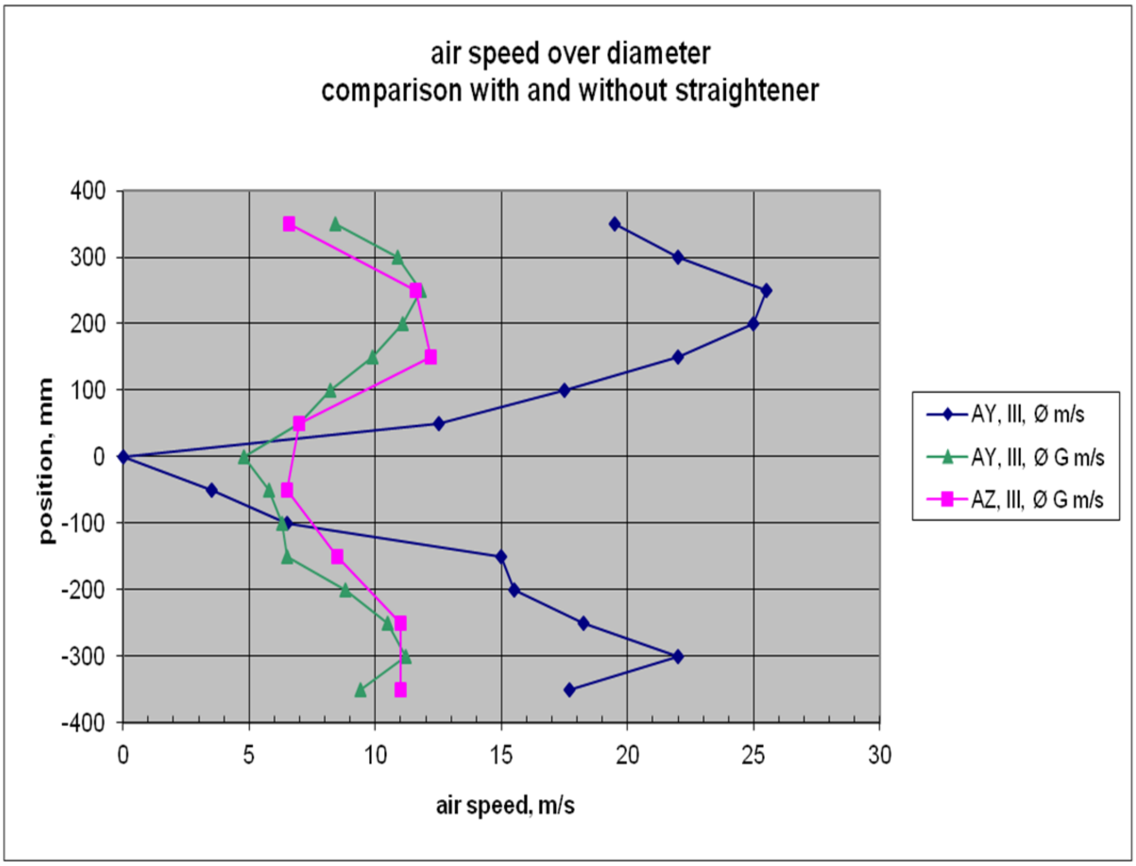


Chart 2 Fan air speed dispersion

4.3 Preparing the car

The car was prepared by installing one temperature sensor to the inlet and the outlet, to measure the different air temperatures. The car already had two other temperature sensors installed in the cooling system to measure the coolant temperatures in the cooling system. Also all the different modifications were tried on the car so that they would fit there and the tests could be completed without any problems.

4.4 Aerodynamic changes

The next chapters will go through the different modifications that were made for the cooling duct. The main points for choosing the different modifications were the ease of manufacturing, the ease of installation and the need of modifications to the actual cooling duct of the car. Last one was the most important requirement because the team didn't allow any modifications to be done to the body. The car had to be kept in show-room condition.

4.4.1 Theory

In this chapter the focus is on giving the basic information on what are the preferences in a cooling duct, when considering the aerodynamically ideal cooling duct.

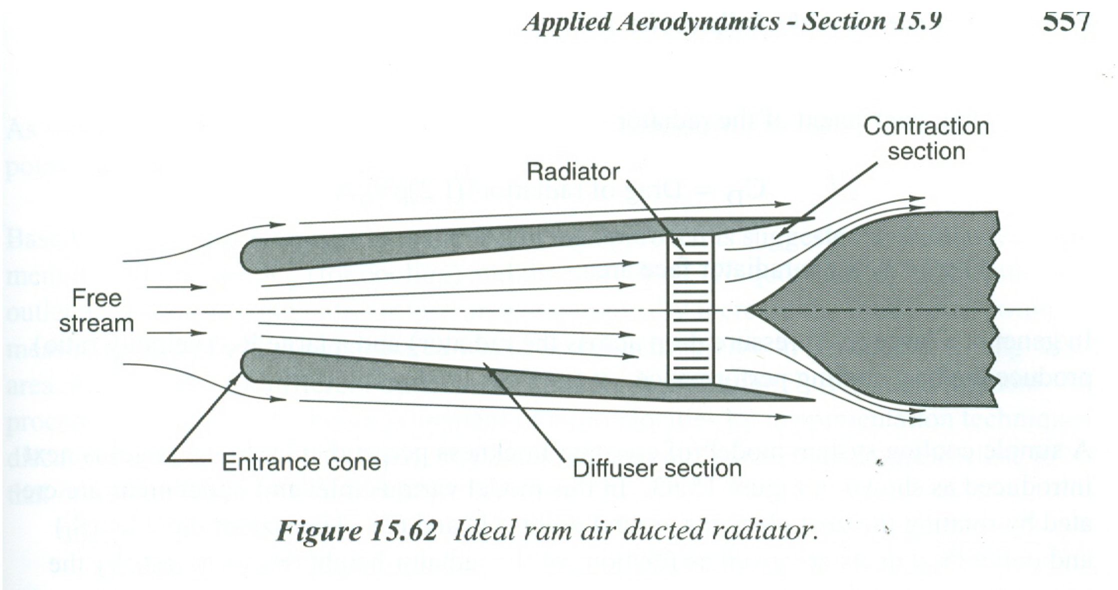


Figure 3 Ideal ram air duct /05/

"1. The entrance cone has well-rounded leading edges. This reduces the possibility of separation in the duct when it is at a modest yaw/pitch angle.

2. The long diffuser with a small diffuser angle (5°) theoretically enables the flow to slow down and build up pressure before the radiator without separation (i.e., in an efficient, reversible manner governed by Bernoulli’s equation). The design could bring the velocity down to a level compatible with the radiator core heat transfer characteristics.

3. The radiator fits tightly in the duct. All of the incoming air must go through the radiator

4. The contraction section aft of the radiator speeds up the air to near free-stream velocity, so it can be ejected with minimum loss. This process is assisted by the heat energy added to the air by the radiator aircraft may actually generate a bit of thrust." /05. p. 557/

In the real-life some of these attributes are impossible to achieve. Firstly what limits the use of rule number two is the fact that the space inside the car is limited. So it's vital to make the duct more compact. The same applies to the outlet side too.

Rule number 3 suggests that the radiator should be installed so that all of the air can go through it. Of course that should be the case, because otherwise the cooling power is reduced. But in real-life the space puts limits for the mounting of the cooler, so not all of the air can go through the radiator horizontally. In the Stallardo the radiator had to be tilted so that the radiator can fit inside the side pod. This inevitably leads to the fact that not all of the air can go through the radiator uniformly, also introducing a small drag increase. Lastly the bigger inlet area doesn't mean that the cooling power would increase. Actually the opposite is true to some extent. In the later chapters this will be dealt in more detail.

4.4.2 Concept 1

The first idea of improving the cooling power and the air flow to the radiator was to get rid of the sharp edges in the inlet. The idea behind this is that the sharp edges create turbulent air flow and also a low pressure zone. When the air hits the edge it doesn't flow smoothly over it, instead it breaks away from the surface, 'thus creating vortices. The same effect happens in the edges of a truck trailer. When the edge is round the air doesn't break away so easily from the surface, instead it follows the shape of the surface. The important point here is that the air flow stays on the surface, 'thus giving a laminar flow of air to the radiator. Also making the edges round the drag of the car would be reduced. The following picture shows the CAS-drawing of the first concept.

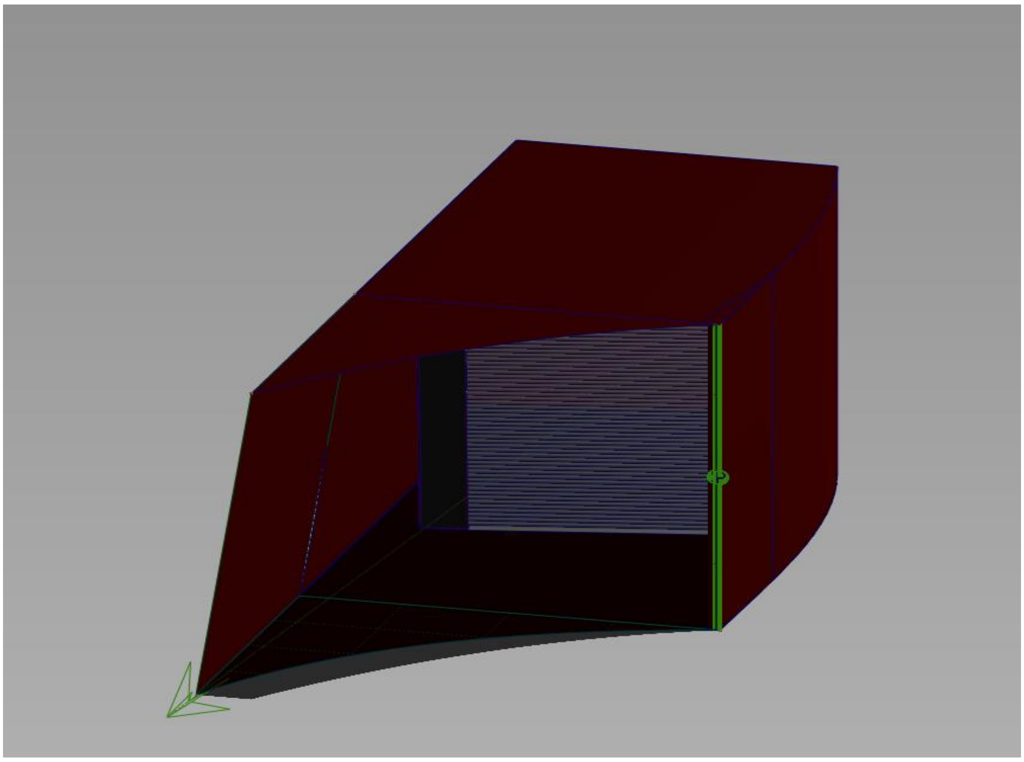


Figure 4 CAS drawing of Concept 1

The idea was to change the shape of the left side of the inlet. The edge became round and the whole left side was extended slightly. This concept also utilizes the biggest inlet area of all of the concepts. The area was around 900 cm^2. According to one theory this should be the best for the Formula Student car. Why? This way the radiator gets as much air as possible. Also this concept doesn't change the pressures and the flow speeds. The average speed of the formula student car is ~60 km/h, so the airspeed is relatively slow. So trying not to introduce more possible losses, this concept was chosen to be used in the test. But as mentioned before the bigger inlet is not always the best. This will be explained later.

Why weren't the upper and lower edges modified too? One reason for it was that manufacturing the parts would have been way too complicated and time consuming. Also modifying the original parts was out of the question, so the only option would have been to build the whole side of the car. This then would have made the baseline test unusable. Aerodynamically thinking changing the upper edge would have been useful.

As a side note this kind of design is the most used in the formula student cars! The next picture shows the actual part installed in to the car. Small changes can be noticed.

Figure 5 The actual part installed to the car

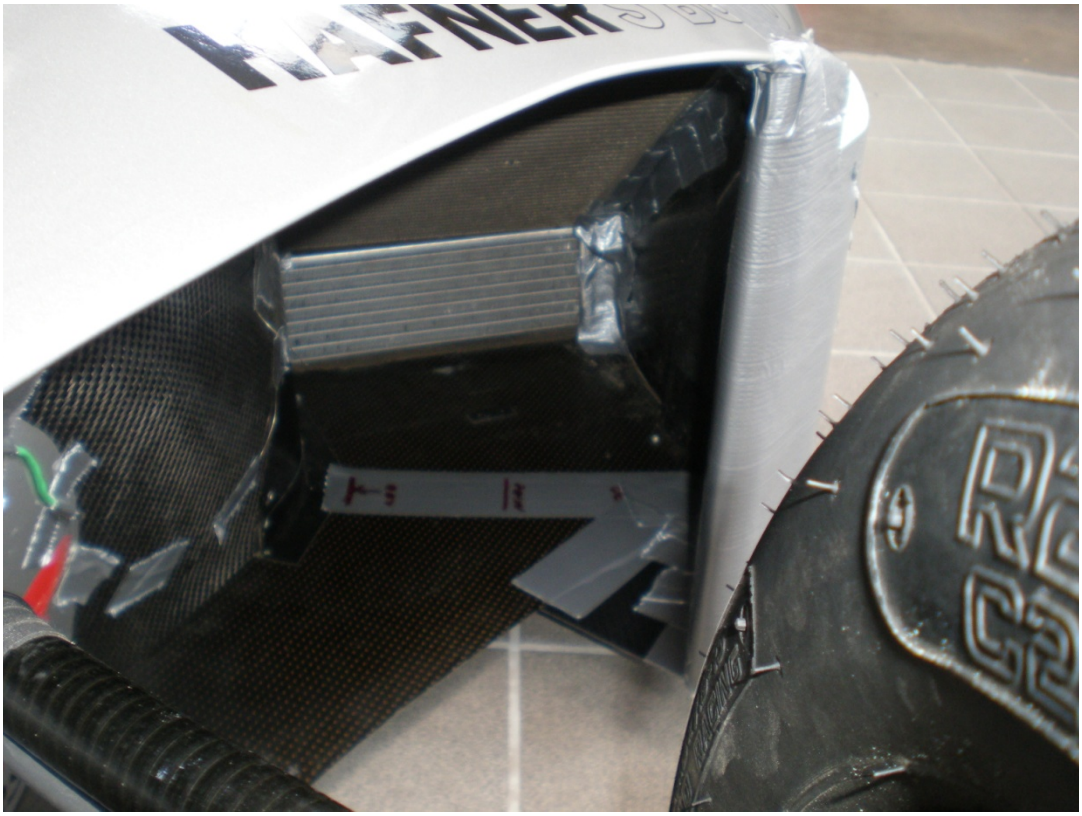


Figure 6 The same modification from the front

The dimensions for the inlet area are width 23cm and height 33cm. So the inlet area was reduced to roughly 750 cm^2. If comparing this to the picture on the page before, few changes are noticeable. During the manufacturing of the part the design had to be changed. The part changed so that the extension was longer than planned and the inlet size was slightly decreased. Also the floorpan was extended to make the inlet duct smoother. Why was this done?

The biggest problem in order to get a laminar air flow to the radiator is the tire, which rotating in front of the inlet and also the suspension components disturbs the air flow. As its well known, a rotating tire produces highly turbulent air behind the tire and as the inlet is exactly behind the tire, all of the turbulent air goes directly inside the cooling duct. Also the air after these suspension elements will follow the same pattern. Aim was to get as laminar flow as possible inside the cooling duct, so actions were needed in order to reduce the turbulent air coming from the tire and the suspension elements. That’s why the part was tilted slightly towards the body, in hope for to deflect some of the turbulent air from the tire and getting more laminar air flow to the inlet. This is noticeable in the picture were the duct is photographed from the front. In that picture the floorpan is not yet extended. Why the area where the part connects to the actual body made smooth. Well the external flow was not an aim in this project, so time saving reasons it was left as it was.

4.4.3 Concept 2

The next concept for improving the air flow was the diverging-converging type. This one is used in almost all of the modern race cars, no matter open- or closed-wheel racer car. This design changes the speed and the pressure of the incoming air. The basic principle behind this is to slow the air in front of the radiator and transform the dynamic pressure to static pressure, raising the static pressure at the same time. This type of duct is usually made long, so that the air has time to settle. In the theory chapter this concept was illustrated.

The diffuser type duct has to be designed to a specific speed and it will work best at that speed. Usually this speed is the top speed of the car, because the need for cooling power then is the most highest. And also race cars that utilize this kind of duct don't have cooling fans. That makes the design even more demanding. Why does the duct have to be design for a specific speed? When the car is travelling at the designed speed and the inlet is matched to this speed, constant desired flow is established. Of course the cooling power is also highest at that speed. When travelling under the design speed the area for the inlet is too small. "In this case the capture area A_∞ is larger than the inlet area A_i and the flow must accelerate ahead of the inlet to meet the cooling requirements." /06. p.217/ This may also lead to a internal separation and that then will increase the drag introduced by the cooling system. The Formula Student car doesn't drive top speeds that often on the track, so the inlet can be designed for a speed that is lower than the top speed. So for example the inlet could be designed for a speed slightly higher than the average speed and when the situation occurs when the speed is lower than average, the cooling fan can be used to accelerate the air to meet the cooling requirements.

Here is the equation to calculate the required area:

$$m = \rho * v * A$$

Where:

m = the required mass flow rate (from the radiator performance charts)

ρ = the density of the air

v = in this case the mean air speed

A = the area of the inlet

However this equation assumes that the flow to the duct is laminar which might not be the case in real life. And also this doesn't take into consideration the drag and the pressure drop in the radiator. There are more precise equations to calculate those, but this will give a result that is close enough to be useful when designing a cooling duct.

Then the situation when the car is travelling faster than the designed speed. "At higher speeds the inlet (with area A_i) will be too large and only a fraction of the incoming streamlines can enter the intake. This results in a spillage, or some local outer flow separations." [06. p.217] When outer flow separation occur this will increase the drag of the vehicle. Even though aerodynamic drag doesn't play a big role in the Formula Student cars, still taking these things into consideration will do no harm for the car. No matter how small the improvement, it's still an improvement. From these two situations described earlier, the first one is the more important one that has to be taken in to consideration in the design phase. The next picture illustrates the actual part in the car.

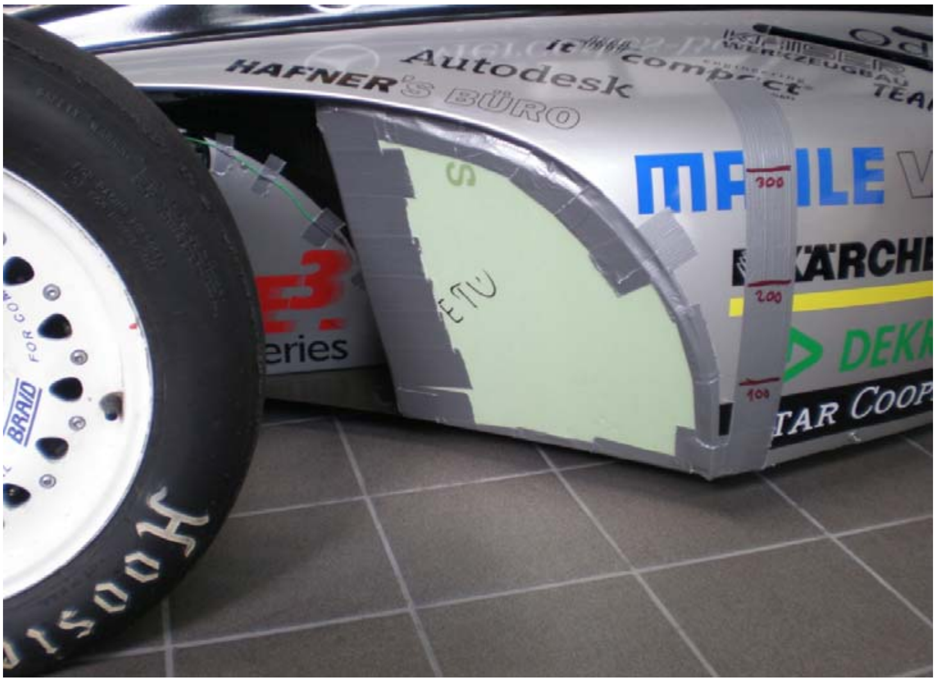


Figure 7 Concept 2 pictured from the side

The first noticeable thing is the inlet area that is drastically reduced. Width of the inlet opening is from 8,5 cm to 13 cm. The height remained the same. This gives the inlet an area of roughly 350 cm^2. Unfortunately this is far from the optimal and the whole duct after the inlet is not how it really should be. The big problem was trying to get information about the installed radiator, example getting performance numbers was impossible. And for the small additional radiator these numbers didn't even exist. Because of this the calculations for the exact inlet area for the designed speed were impossible. So the area used in the test was just a random area, just to study the effects of making the inlet smaller and seeing if this concept actually works in a Formula Student car. For the actual duct the inlet area has to be calculated precisely. In the following pictures are these two concepts illustrated and it is noticeable how much the inlet actually changed. Even from the concept 1 the inlet size reduced more than 55%.

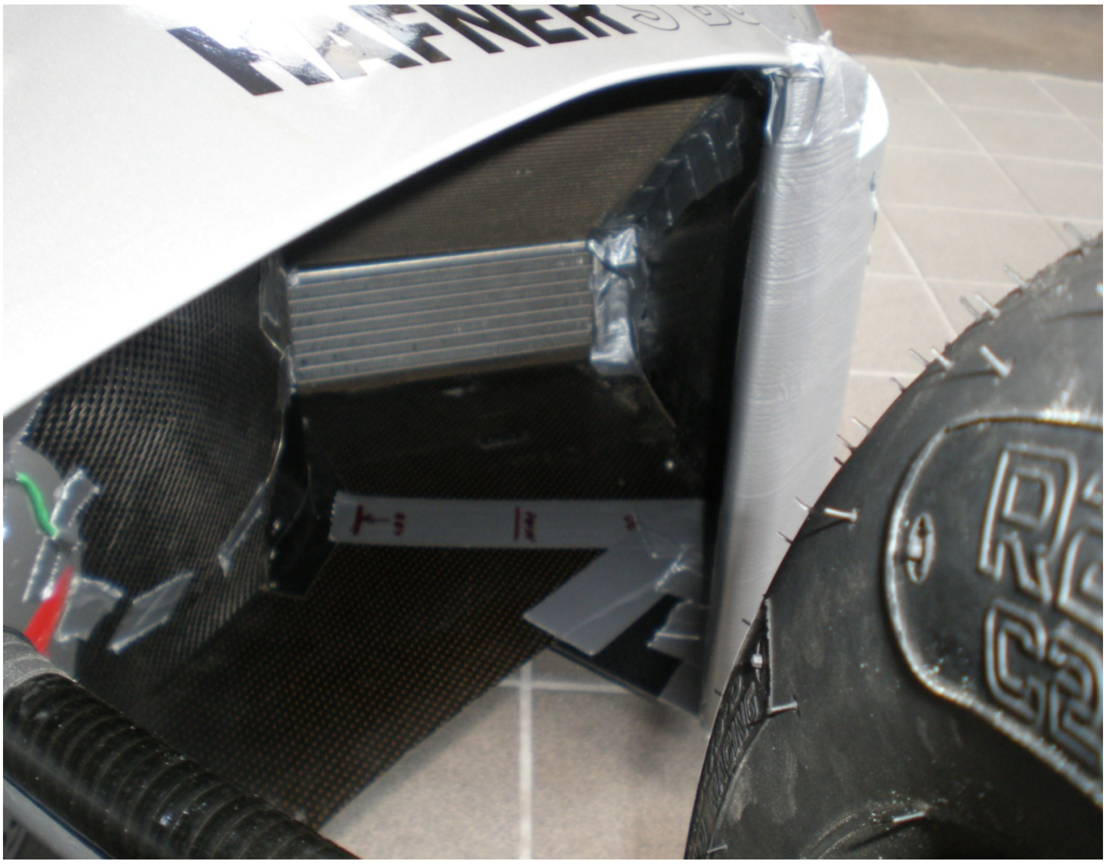


Figure 8 Concept 1 front view

Figure 9 Concept 2 front view

4.4.4 Barge board

A barge board or a turning vane is relatively new device in the aerodynamic world. The barge boards were introduced in 1990's. Back then they were quite simple devices but since then they have evolved heavily. In the beginning they were like our barge board in the picture, now they are included in the body more and have really complex shapes to full fill their function in various parts of the front section. In the next picture the actual part is installed to the car. The picture was taken while the test was being done.

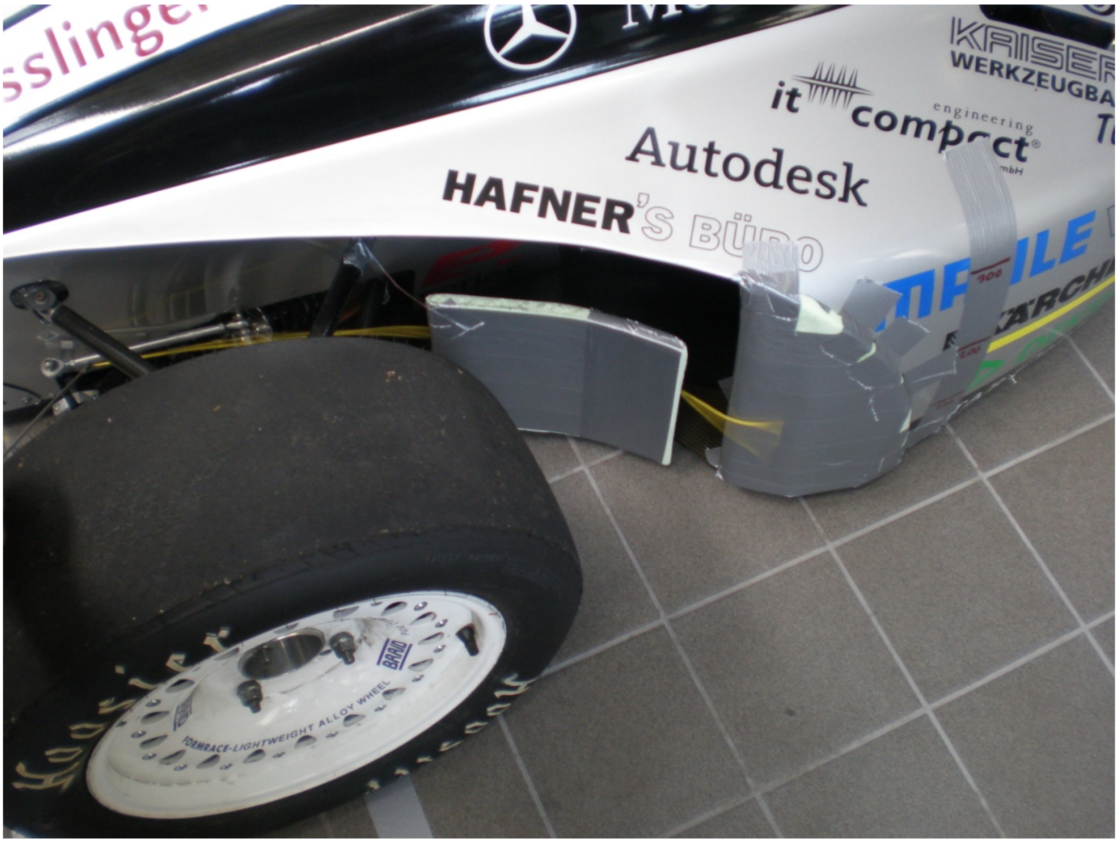


Figure 10 Barge board with concept 1

One of its main jobs is to work as a flow conditioner. What a flow conditioner then does, is to guide and smoothen the air flow. It also prevents the highly turbulent air from the tires from entering the cooling duct. Now most of the F1 teams use it to introduce vortices that go under the car, by doing so they lower the air pressure under the car, 'thus making more downforce. How to make these vortices is by having really sharp edges on the bargeboards. So when the air gets close to the edge it tries to roll around it, but because the barge board ends, the air continues with a rolling motion. Of course at this point the flow is turbulent and the pressure drops. This is the same thing that happens in the airplane wing tips. It can be seen really clearly when looking at a plane when it's landing on a rainy day. For the test though the barge board was designed to guide the air flow. Introducing vortices was not planned.

Measurements for the barge board are: width 25cm, height 23cm, thickness 1,5cm

The height is roughly $^{2}/_{3}$ of the height of the duct, so it should protect the inlet quite effectively. Width was chosen so that it would reach from the edge of concept 1 close to the body of the car. This part was manufactured first so when the actual concept 1 was changed slightly, the barge board still remained as it was. Mainly due to time, changes weren’t made to the part. When the barge board was installed in the car, small gap was left between the barge board and the edge of the inlet. Also by extending the floorpan the barge board had to be installed in a bigger angle in a respect to the body. The model for the leading- and trailing edge of the barge board was taken from air-plane wings. The leading edge was done so that the air flow wouldn't separate from the surface and the trailing edge then was shaped so that the flow would leave the sur-face without much of a disturbance. As stated earlier no vortices wanted to be created.

4.4.5 The suspension modifications

With the suspension components the aim was to do what all of the Formula 1 teams are doing and that is to have wing shaped ‘’bars’’, instead of just normal round bars. In Formula 1 cars all of the suspension components are shaped so that they wouldn’t produce a turbulent flow behind them. A normal round bar introduces a wake after the bar and that wake is heavily turbulent, somewhat like the wake what the tire introduc-es. These components are right in front of the cooling duct inlet so modifying these parts is a good way of reducing this turbulent air flow from entering the duct. The following picture shows the actual parts in the car.

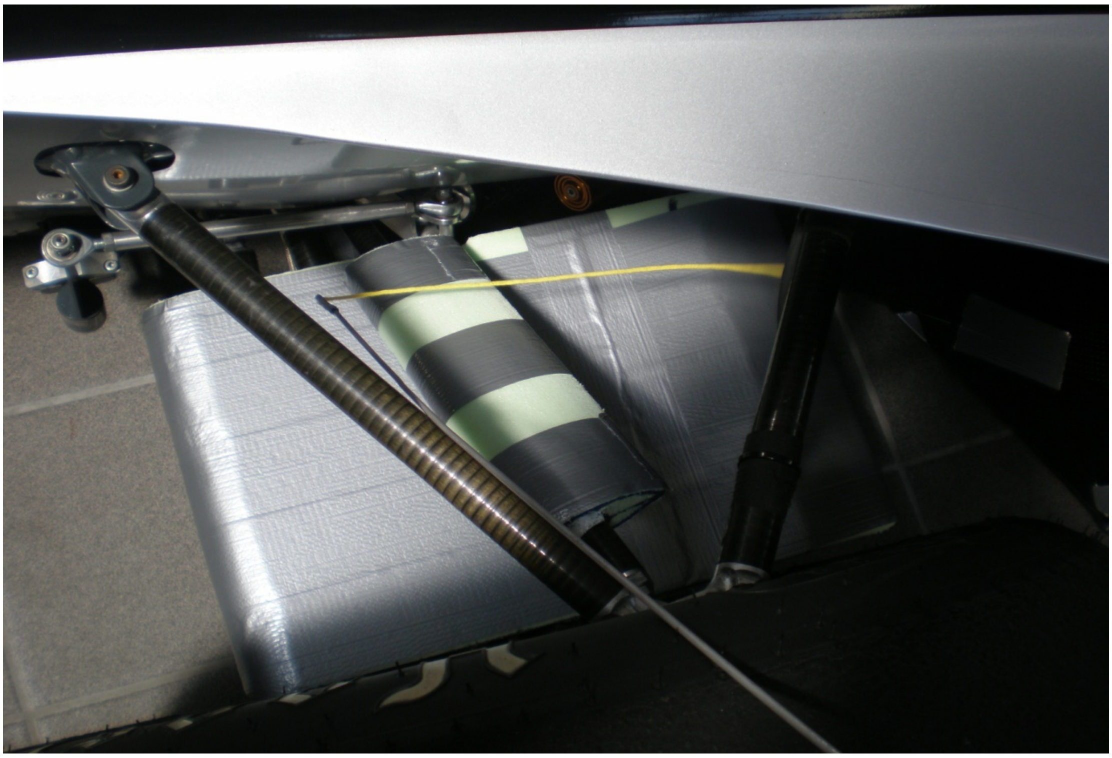


Figure 11 The suspension modifications

The first noticeable thing is that the part that is covering the lower bars is not really a real world example. With this modification tires are not able to turn at all. This fact was known during the build of this part and there is a good reason why this was done like the way it was. The first part that was manufactured was the middle one. Building that was quite difficult because the thickness had to be really small, but making it really thin the part became fragile. So after finishing the middle part, it was decided to just cover the lower bars with one part. That made the manufacturing easier and also it cut the manufacturing time in a fraction of what it would have been. In theory having 3 individual parts wouldn't be a good idea unless all of the parts are designed perfectly. If any of those parts are defect in any way, it will then affect the others too. Now when there is a constant surface that the air follows and it doesn't have to separate at any point, it reduces the risk of creating a turbulent flow between the parts. In reality this of course wouldn't work, because of the reason stated earlier, but this is just a simplification to see does it have any effect on the air flow. In the next picture actual Formula 1 suspension components can be seen. It's quite noticeable how different they are from the parts that were installed in the Stallardo.

Figure 12 Formula 1 suspension elements /11/

In the following chapters I will explain the results of the test, with text and pictures.

5 RESULTS

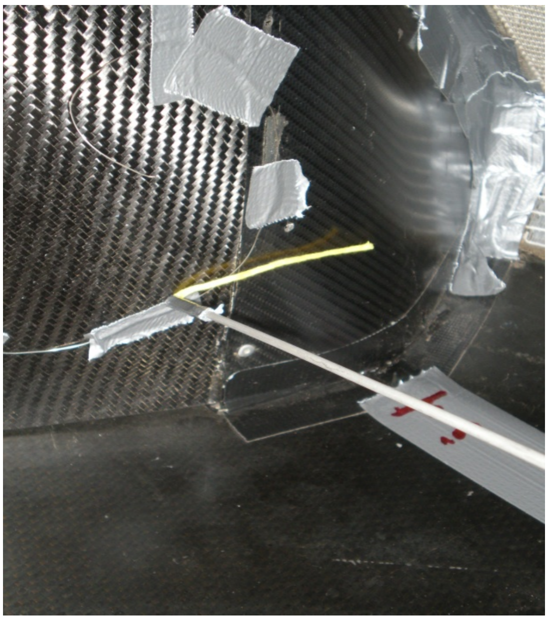
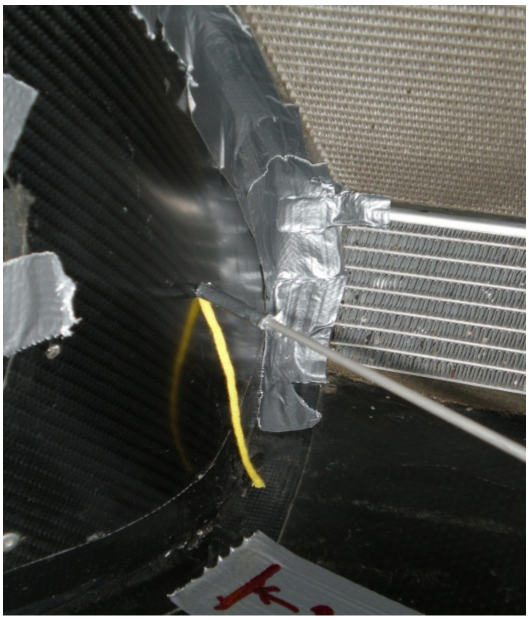
The results will be presented in the order in which the tests were made, starting with the benchmark and ending with the suspension modifications. In the results will be the valuation of the actual air flow and then the cooling performance, what was then measured with the Specific Dissipation (SD) method. Every modification will be dealt individually and lastly are the air speed measurements.

5.1 Benchmark

Figure 13 The original duct during baseline test

Here is a picture of the original duct taken during the base line test. The most noticeable thing is that the air is flowing out of the duct. The whole inlet area is very turbulent and the air is going all over the place. Finding a laminar flow in any single point of the inlet was impossible. This result was really shocking because this wasn't expected to happen at all, especially when this kind of design is used so widely among the Formula Student community. It was clear at this point that this kind of design was far from being the ideal and even when the fan was turned on it had no effect on the air flow. Most likely the tire was affecting heavily on the air flow. When the tire is not rotating the pressure diagram is quite different from if the tire was rotating. Now the tire was creating a huge low pressure behind the tire and this low pressure was sucking the air out from the cooling duct. Also the pressure behind the radiator wasn't low enough to help to draw the air through the radiator. But because the pressure measurements were not done during the test, it is impossible to validate this theory. And perhaps if the tire would have been rotating the result might have been different.

Another thing that was noticed was that the air wanted to follow the same path as the upper side of the body. So not only the air wanted to come out of the duct it also didn't want to go in. This was quite strange because looking at the duct it would be obvious that after the suspension elements (even if the air is turbulent) it would go straight into the inlet. Perhaps the low pressure zone that the tire created was affecting to this too. The next two pictures show well how the air flow changes inside the duct just before the radiator.

The picture on the left side shows how the air flows nicely on the side of the duct. That is what should happen inside the duct. Then the picture on the right side shows the same air flow but just 10 cm closer to the radiator surface. Now the flow doesn't even exist anymore. The wool string was just hanging there even though the wind speed was around 5 m/s! So from that the conclusion can be made that the inner side of the duct is not designed right and the flow separates after the ridge. This shows that the inside of the duct really has to be designed so that it's smooth and it doesn't have any sharp angle.

For the benchmark test the SD-values were calculated too. The following chart shows the specific SD-value in comparison to the time. With the fan off the SD-value was about 60 W/K. When the fan was turned on the value was 100W/K. The fan increases the cooling power by ~60%. This shows how much the fan has an effect in drawing the air through the radiator and 'thus improving the cooling. The rise after the 100 second is the point when the fan was turned on.

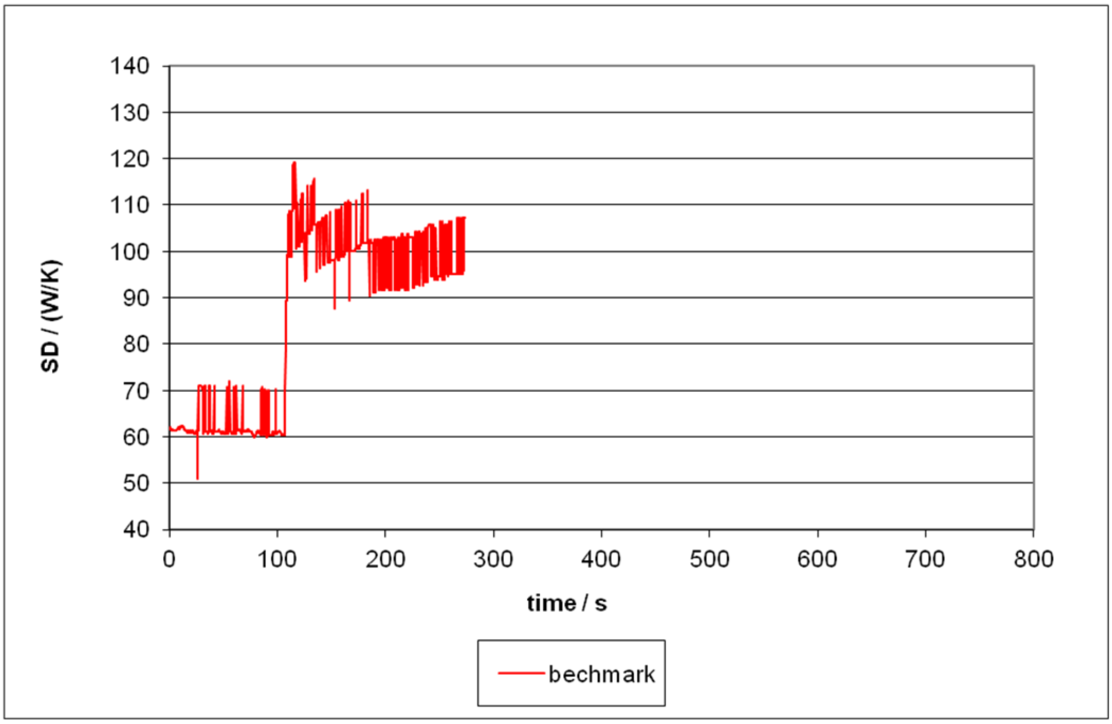


Chart 3 The SD-value from the benchmark test

5.2 The Concept 1

There was some amount of luck involved with the design of concept 1. How the air would actually move was not know before the test, so tilting the part slightly was a really good decision. What was noticed in the first test was that the air wanted to flow parallel to the upper edge, so installing this ‘’scoop’’ to the edge ended up working well. The following picture shows how the concept improved the air flow to the cooling duct.

Figure 14 Concept 1

Of course the flow still follows the same path as before but now the added part on the side helped to get more air in. The air flow inside the cooling duct was still highly turbulent. This was expected to happen because basically nothing was done for the inside part of the cooling duct. Perhaps some improvements happen to the air flow close to the outer edge of the cooling duct. Now the air had more time to settle because the outer side was longer than before. Although not much happened to the air flow, the SD-value took a turn for the better. It increased from the benchmark result of 60W/K to 75 W/K. This was a 25% increase in the cooling power. This shows how this relatively simple part increases the cooling power quite significantly. In the following chart are the results shown.

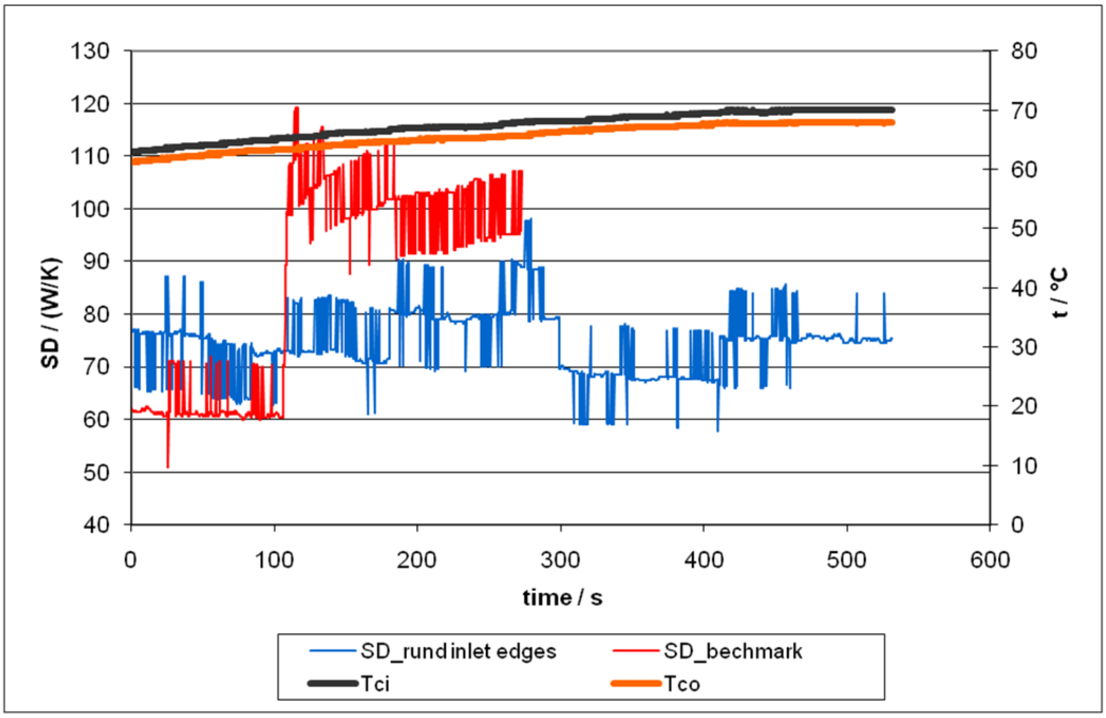


Chart 4 The SD-values of the benchmark and the concept 1

5.3 The Concept 1 with barge board

By installing the barge board it was expected to get better cooling power and better quality air flow inside the duct. In the next picture the flow of the air is well noticeable by looking how the wool string flows. It is obvious that the air flow is far from being good. The flow follows the inner side of the barge board and then continues forward, pass the small opening between the barge board and the edge of the inlet. The choice for the angle and the position for the barge board wasn't good. The lack of solid information about the barge board made any calculations regarding the positioning or the angle impossible. So it was decided to installing it so that it follows the upper edge of the side pod.

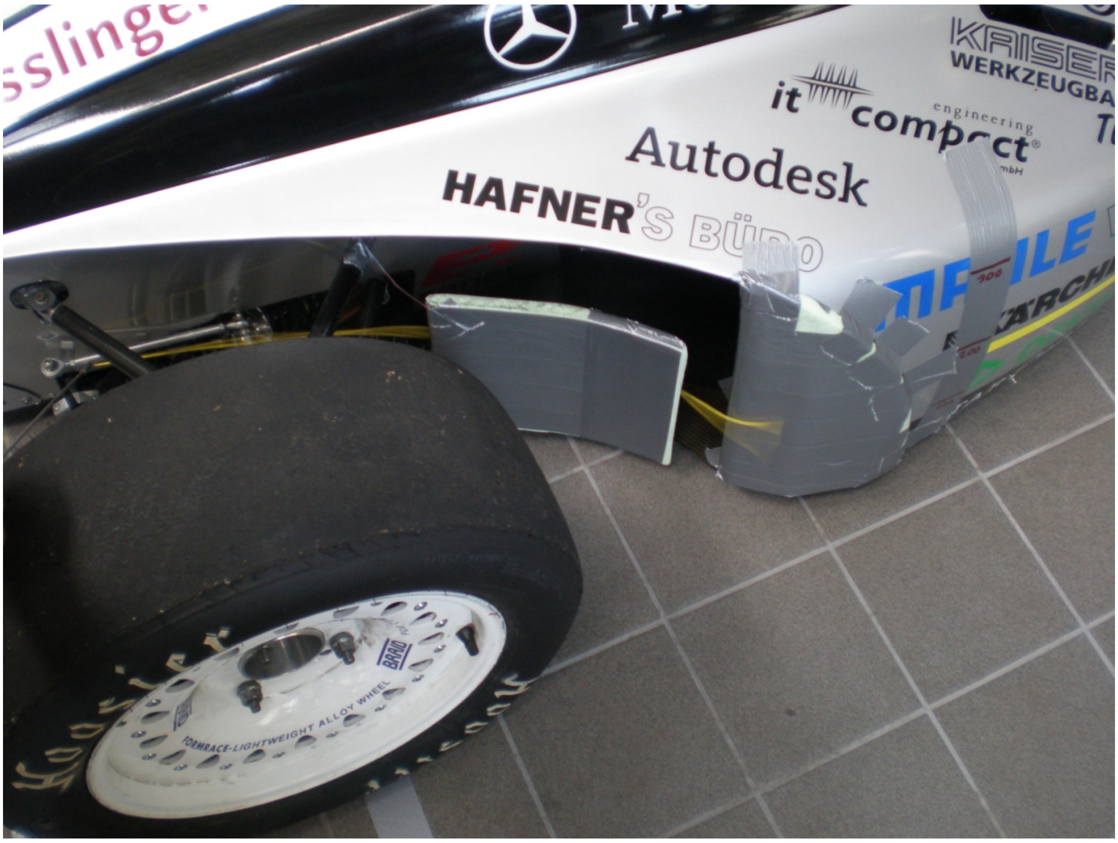


Figure 15 The air flow with concept 1 and the barge board

Better position for it would have been closer to the side of the body and closer to the suspension elements. The angle should have been somewhere around 10-15^{o}, not the ~45^{o} that was used during the test. Unfortunately all this information was found out after the test. So because all of this the actual cooling power reduced! But what was better was the flow inside the duct. So what was done was getting less air in, but made it more stable. The turbulence was still present as it was with all of the modifications, but it was toned down a bit. The picture below shows the flow inside. Most of the air from the suspension arms is blocked by the barge board and more air from the side of the body is directed in.

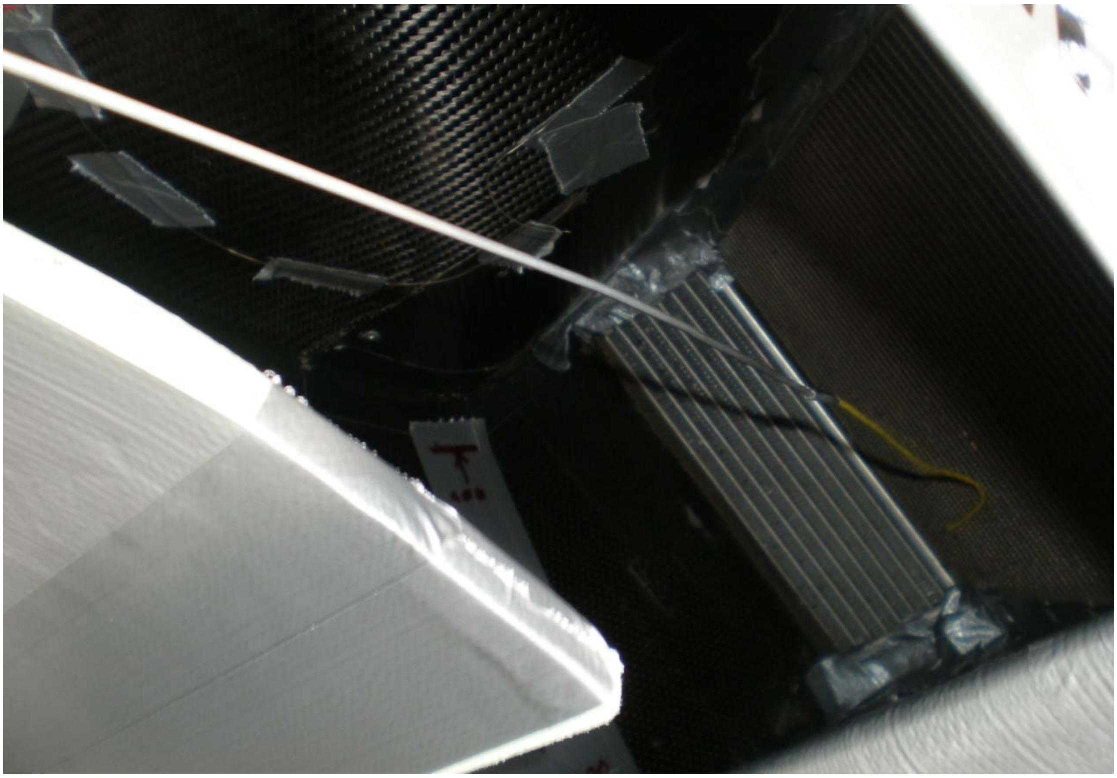


Figure 16 The air flow inside the duct with the concept 1 & barge board

Like already mentioned with this addition the expectance was that the cooling power would increase from the concept 1. But what happened was that comparing to the SD-values of the concept 1 the values decreased. Although still being better than the benchmark result by 17%. The values were 70W/K with the fan off and with the fan on it rose to 105W/K. In the next chart the SD-values are shown.

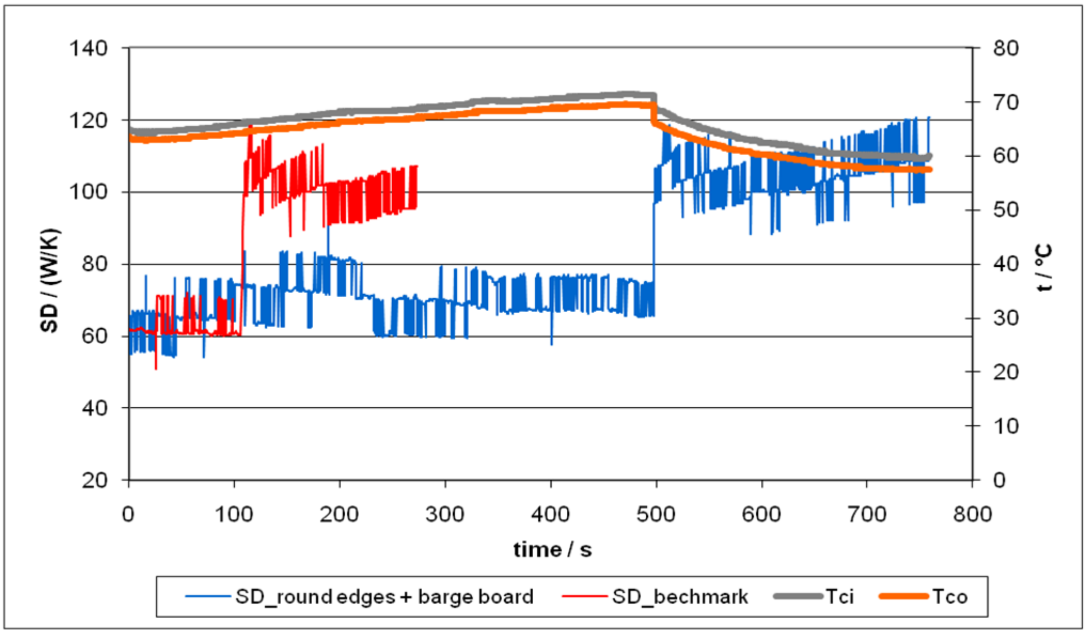


Chart 5 The SD-values with concept 1 & barge board

5.4 The Concept 2

Next the concept 2 that proved to be the most interesting of all of the different concepts. In the next picture it's quite noticeable how much the inlet lost its area. The actual size of the inlet was ~350 cm^2. Comparing that to the original area of ~950 cm^2, the difference is truly massive. Even the team had their doubts on this and was concerned that the engine might boil. According to the books and math this would be the best and provide significantly better cooling power than the original one.

What was immediately noticeable when the wool string was positioned in the inlet was how much better the air flow was. The next picture shows how the air flow goes straight into the duct. This was exactly what the theory suggested what would happen. Now because the air that was going in is taken from the side of the body, it's not as turbulent as if it would have been taken from the free stream after the suspension components. But this concept had its own problems too. The holes in the body are acting like NACA-duct sucking air in and this is not good regarding the drag. The holes housed the shock and the spring, and also the suspension arms mounting points, so it was impossible to do anything for them. The lower picture shows the problem.

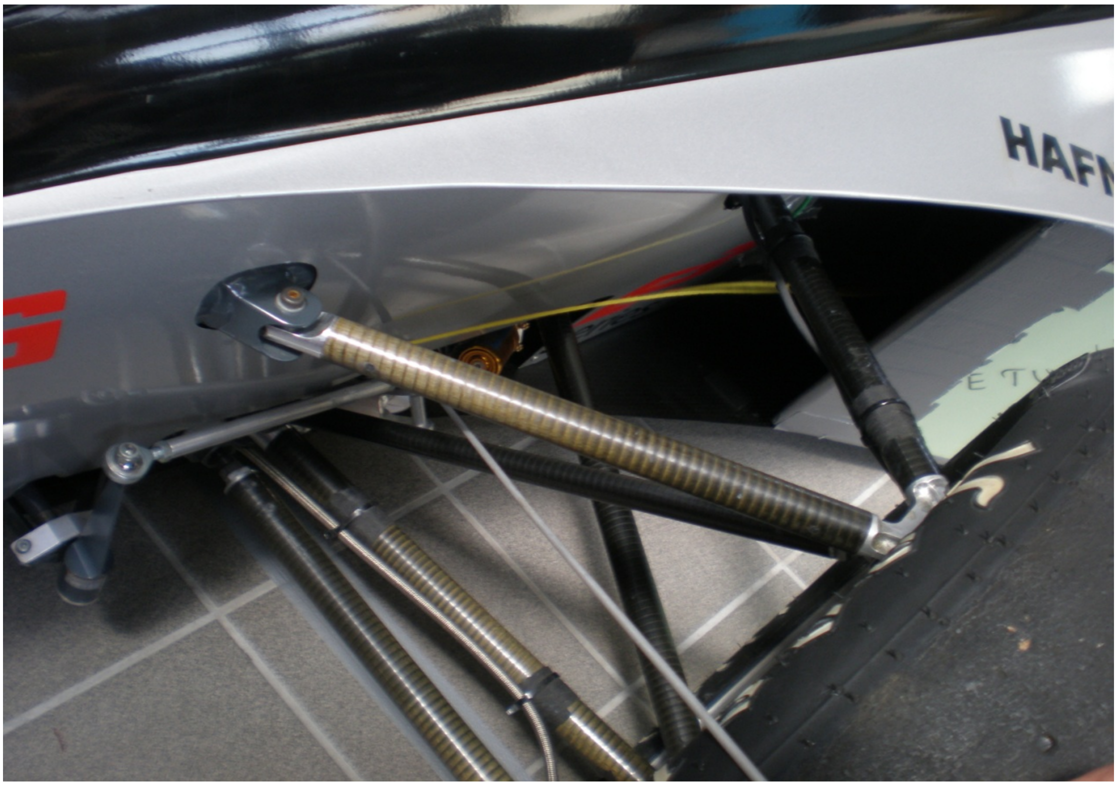


Figure 17 The air flow with the concept 2

Figure 18 The air flow entering the suspension arm housing

The air flow inside is flowing nicely towards the radiator and because most of the inlet is closed, it cannot escape. It was hard to study the flow inside the duct but the first picture above is showing quite well how the air is entering the duct. And as mentioned before, the inlet area was not the optimum nor was the actual shape of the duct. So if the cooling duct would be made like it’s supposed to be, the results would be even better. Also combining this with a really good outlet that provides sufficient low pressure behind the radiator, then the flow in front will be enhanced even more. Unfortunately different outlets were not tested with this concept. The results from those tests would have been very valuable for the future design of the duct. The nest picture shows the air flow inside.

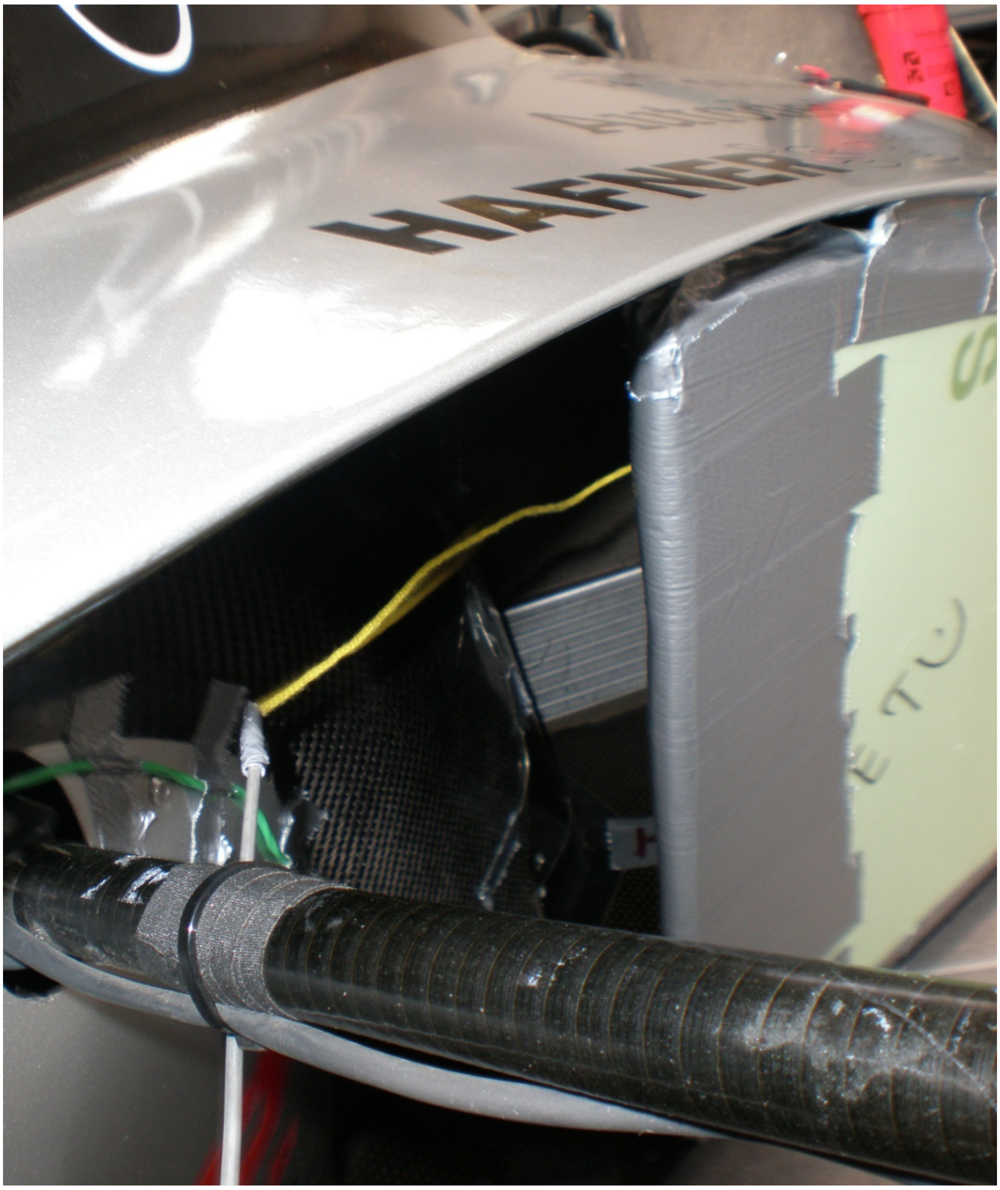


Figure 19 The air flow inside the duct with concept 2

The SD-results with the concept 2 proved to be the best. From the benchmark test the SD-value rose roughly by 30% to 80W/K with the fan turned off. When the fan was turned on the result was still better than the benchmark but not with such a big margin. The cooling power increased only by 10%. Comparing this to the benchmark test increase of 60%, it's quite low. Most likely the system as a whole was reaching its limits and the cooling power couldn't increase over 120 W/K. The inlet part was working much better than before and this resulted in the elevated SD-value. Now the outlet part with the cooling fan wasn't up to par, so it was actually limiting the cooling power. Actually the outlet for the car was designed in a hurry, so it was far from being the optimal.

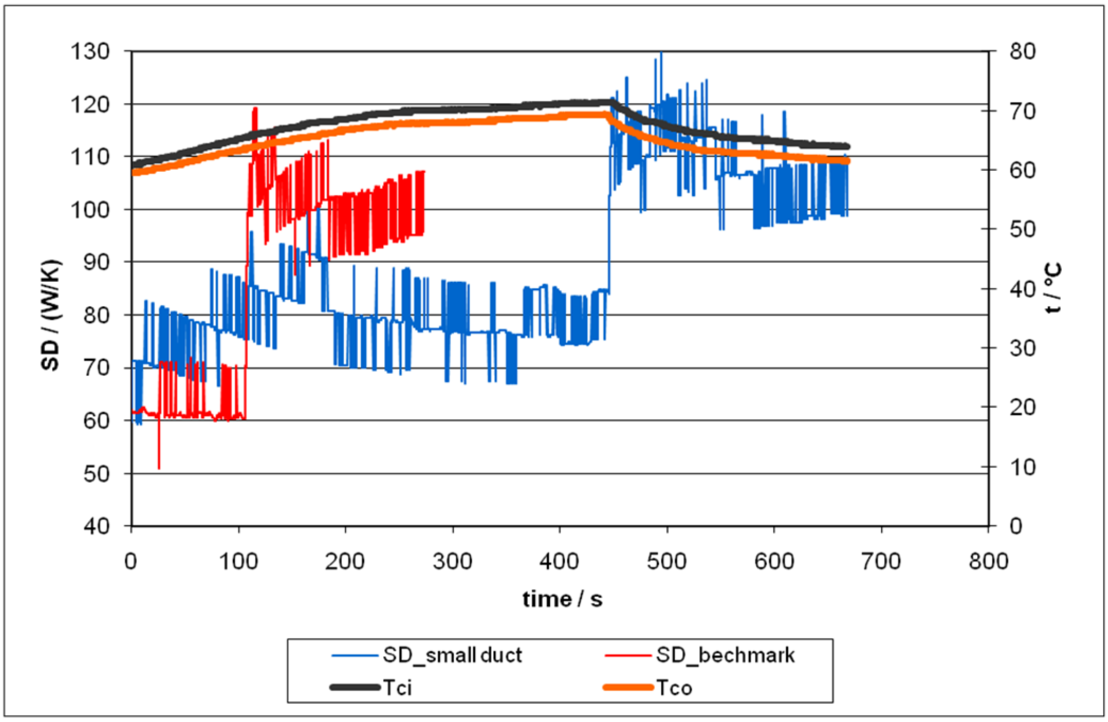


Chart 6 The SD-value with concept 2

5.5 Suspension modifications

Testing of these parts was decided to be done alone, separate from any of the modifications. The main reason being that it was a simplification, and the other one being that the space available to fit all of the parts at one was impossible (referring to the concepts 1 & 2). So the test was done just to see if it's possible to improve the air flow by doing this streamlining of the parts. The next picture illustrates how the wool string lines up, the familiar path is still there, but this time with less turbulence right after the suspension arms.

Figure 20 The air flow over the suspension modifications

It’s quite clear that the air flow is much better than when there was only the round bar. The air flow doesn’t get detached from the surface and so it stays laminar for longer. But because the airfoil doesn’t reach all the way to the end and it has this sharp edge, it produces wakes behind the airfoil and these wakes results in turbulent air flow. The part that was installed to cover the lower suspension components, didn’t work as well as the one above. Because the bars where really thick (Ø20mm) the part ended up being quite thick, too thick in fact to work perfectly. It did improve the flow, but not as much as hoped for. Most likely besides the thickness, the leading edge is too blunt and the trailing edge angle isn’t small enough and this leads to a flow separation.

Although the parts weren’t perfect it still produced results that were positive. In Formula 1 nobody uses round bars for the suspension arms that are in the airstream. So it does help if the parts are constructed so that they are flat as possible, 'thus reducing the turbulent wake behind them. How difficult they are to produce that is another thing, but that’s for the team to decide if it’s worth doing.

5.6 Airspeed measurements

The airspeed measurements were done with a vane anemometer at the inlet of the cooling duct. The results that were achieved are not absolute, because the measurements proved to be problematic. The main problem being that having no rigid measuring grid in front of the inlet, it was difficult to keep the vane anemometer at a specific point the required time, so too much error can be in the results. Also the yaw angle of the vane anemometer had to be kept close to 0°. Because of this reason the result can be considered just a snapshot of the inlet speeds. But in any case it still confirms the SD-evaluation method being true.

The following chart shows the different speeds that were calculated from the values that were established with the measurements. Even thought the changes are marginal it still shows how the speeds are slightly higher with the modifications that were done. Perhaps if the airspeed would have been higher during the test, more noticeable results might have been established.

	airspeeds	m/s
	fan off	fan on
Bechmark	0,6	1,2
Concept 1: Round inlet	0,7	---
Concept 1: Round inlet and a barge board	0,7	1,2
Concept 2: Small inlet	0,8	1,3

Chart 7 Calculated airspeeds

6 CONCLUSION

In the beginning of the thesis I laid down three attributes through which I can assess if the project was successful or not. These were the cooling power, the air flow and the pressure losses/drag. The cooling power, which was measured with the SD-value, increased in every single modification comparing to the baseline value. With some of the modifications the increase wasn’t much, but with the most important one, concept 2, the increase was significant. Over 30% increase in the cooling performance without doing anything to the actual hardware. This is a great improvement! If the cooling duct would be designed to the specific speed the increase would be even more. Then the radiator could be made and designed to utilize the new duct more efficiently. The weight of the radiator could then be reduced, which is for the Formula Student car always a positive thing.

Next the air flow. Now this one is a hard to assess, but just by comparing how the wool string moved in the air stream with and without the modifications, improvements can be noticed. The turbulence in the duct was reduced slightly and the air was flowing more to the radiator, instead of flowing out of the duct. Any substantial evidence that the air flow truly improved, I can’t show. But judging by the cooling performance the air flow has to be better than in the original design.

Lastly the pressure losses and possible drag reduction. Like already mentioned, any of the pressure measurements weren't performed so I can't say with complete confidence that the dynamic pressure changed to static pressure, and also that the pressure rose. But again judging by the increase in the cooling performance and also looking at the speed measurements portrayed earlier, I can draw the conclusion that there was a small pressure rise and that most likely the pressure was transformed to static pressure. When we had done the calculations for the pressures, the rise in the pressure was extremely small, but a rise none the less. Because the speeds in which the car operates are relatively slow the rise in the pressure was expected to be quite small. Then most likely by installing these parts to the cooling duct, the drag was reduced slightly. This conclusion can be made by studying how the air flows in the cooling duct. Unfortunately aerodynamic drag is extremely difficult to measure, so concrete results can't be declared.

Comparing these results to the literature, the results follow the same path. What was written in the books was exactly what happened in the actual test. From this we could conclude that the test was a success and done with enough accuracy. For the future I would recommend doing these tests again and evaluating the results that we got. Also it would prove as a test for the Specific Dissipation method, that can it be repeated. Because some amount of error is of course in the results. The main reason for that being human error, during the test and after that while doing the calculations, secondly the errors in the sensors and other measuring devices. And to assess the air flow, the use of smoke would be the best way to do so. The wool string does its job well, but with the smoke it's clearer to see the changes and also it gives the chance to view the whole flow in the cooling duct. For the speed measurements the only way to do it accurately is to do it with the pressure measurements. The small vane-anemometer that was used in the tests is too easily affected by the angle in which the air hits the vanes. This makes the results vary too much.

Another suggestion for the future is to take the concepts 1 and 2, and further refine them. Calculate the exact area needed for the average speed and then design it so that it fits the body perfectly. While doing this also looking the outlet side and try to improve the exit flow of the air. The outlet has a huge effect in the air flow. By combining these two the results would be even better. And lastly CFD-simulation would be one tool to assess the design of the cooling duct. CFD-simulation possesses a lots of advantages but the main problem with the CFD is that, to have a really reliable results the model has to very exact. And trying to achieve this takes a lot of time and effort. Also one has to have a hardware that can do all of this. If this hardware is not present, buying one requires a lot of money so it can't be considered a cheap choice to assess the aerodynamics.

7 LITERATURE

/01/ National Automotive Radiator Service Association. Cooling system. WWW document. www.familycar.com/Classroom/Images/Cooling-System.gif. No update information available. Referred 3.9.2011

/02/ Stefan Nasello. 20.4.2011. The Data Acquisition Engineer, Rennstall. An interview

/03/ Eton Yat-Tuen Ng. Vehicle Engine Cooling Systems: Assessment and Improvement of Wind-Tunnel Based Evaluation Methods. RMIT University. 2002.

/04/ Lin, Saunders, Watkins, Mole. Increased Productivity - Use of Specific Dissipation to Evaluate Vehicle Engine Cooling System. Society of Automotive Engineers. PDF document. http://papers.sae.org/970137. 24.2.1997. Referred 4.9.2011

/05/ Milliken, Milliken. Race Car Vehicle Dynamics. Warrendale. Society of Automotive Engineers. 1995

/06/ Katz, Joseph. New Directions in Race Car Aerodynamics. Cambridge. Robert Bentley, INC. 1995

/07/ Christoffersen, Söderblom, Löfdahl. Improving the Cooling Airflow of an Open Wheeled Race Car. Society of Automotive Engineers. PDF document. http://papers.sae.org/2008-01-2995. 4.12.2008. Referred 4.9.2011

/08/ Hucho, Wolf-Heinrich. Aerodynamics of Road Vehicles. Warrendale. Society of Automotive Engineers. 1998

/09/ Siitonen, Capriotti, Schumacher. Cooling System - Aerodynamic Analysis. Rennstall. PDF document. 10.07.2011

/10/ Edwards, Simon. Behr GmbH - Engine Cooling Systems. Lecture PDF document. Secondary source. 12/2010

/11/ Formula1.com. Suspension and steering systems. WWW document. http://www.formula1.com/photos/597x478/manual/suspension01.jpg. No update information available. Referred 2.11.2011