Computational Simulation of the Flow Around a Rim of Competition

Garcia, G.

Department of Mechanical Engineering

Universidade Técnica de Lisboa, Instituto Superior Técnico

Av. Rovisco Pais, 1049-001, Lisboa, Portugal

Abstract

In the world of motor racing, all details are important to gain advantage over the competition. This work has the purpose to improve the performance of the *electric Formula Student*, constructed by students of the *Instituto Superior Técnico*, more specifically to improve the cooling of the brakes and electric motor, which are inside of the rim.

To achieve this goal, a computation method was used. Some rim models have been designed in the software *Solid Works*, and the aerodynamic analysis was made in the *Star-CCM+* program. This analysis focused on the comparison of values of air stream that flows through the interior of the different models analyzed.

In the model that obtained the best results, it was verified a improvement that increased in 30 times the air flow in relation to the original version. This improvement will allow a better performance from the braking system and the traccion system, and increases the lifetime of both mechanical components.

Keywords: Electric Motors, Wheel Rims, Brake Cooling, Computational fluid dynamics, Star CCM+

1. INTRODUCTION

This work is part of a project which purpose is to develop a *Formula Student* with an electric motor. The motor will be placed inside the wheel rims, near the brakes (motor in-wheel). This way, several mechanical components are neglected, such as transmission shafts and differentials. The wheel movement could be made independently, which could allow a better traction control. This way, the efficiency will be improved, because there will be less mechanical losses, and the weight will be lower.

One of the known problems of the electric motors is the decrease of the efficiency with the increase of the temperature, which could damage the motor, as well as the brake system. So we should find a solution to cool both systems. This work will study the hypothesis of having a set of blades within the rim which will function as a ventilator when the vehicle moves and will force the air to flow inside the wheel. It will be done by computational means and will try to find the type of ventilator that induces de maximum air flow. The original wheel can be seen in the next picture:

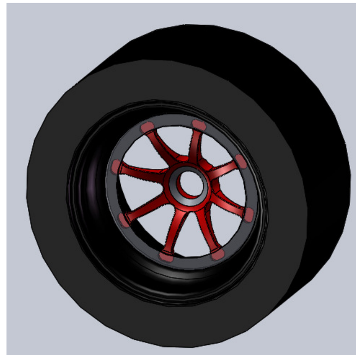

Fig. 1 – Computational model of the original wheel

2. APPLICATIONS

Some develops have already be made so far in the automobile industry. For instance, BMW launched in the late 1980s the model *[E34] M5,* which was one of the first commercial cars that possessed ventilators in the wheels, and they improved in 25% the air flow through the rim. In spite of these improvements, this development required the manufacture of two sets of rims, one for each side of the car, and it was severely criticized by the press due to its looks.

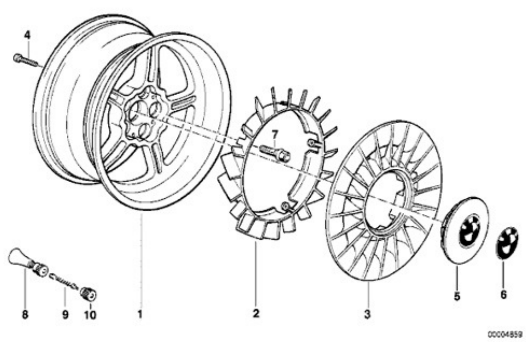

Fig. 2 – Exploded view of its rim

This model wasn't appealing enough to the consumer, so the production was interrupted in the early 90s.

Mercedes also did some developments in this area, and used a set of rims that helped cooling the brakes in its super sports cars, the SLR. Its design is very appealing to the public in general. It's important to combine functionality with looks, and this model was definitely a success. Two sets of rims are also required, as it can be seen in the next picture:

Fig. 3 – Both sets of rims

Most of the inspiration required to do this work was obtained in the *Formula One's* world. In 2007 there were used and developed two different kinds of aerodynamic devices in the wheels. In the front wheel, it was used a rigid device, with a small opening near the brakes, which was entitled rim shields.

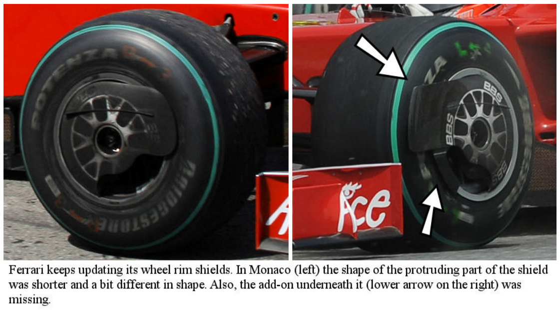

Fig. 4 – Ferrari's front rim shields

The rim shields have a different purpose than the one it was studied. They were designed to reduce the turbulence in the outside of the wheel, and increase the downforce. The rear rim was completely different. There were no static parts, and there could be seen some solutions, depending on the car's constructor, that increased the air flow in the inside of the wheel. For example, in 2009 the Williams used this solution:

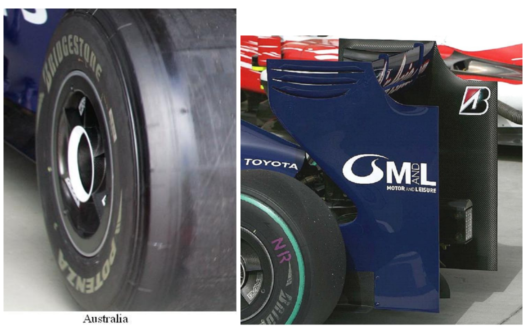

Fig. 5 – Williams' rear rim

Other teams, like Ferrari and McLaren, chose another kind of rims:

Fig. 6 – Ferrari and McLaren's rear rim

The shape of these rims forces the air to flow inside the rim, helping to cool down the brakes. 2009 was the last year that these devices were allowed in the Formula 1.

3. WHEEL MODELLING

The whole vehicle has influence in the flow around the wheel. In the beginning it was thought to use the car in one piece and analyze the flow around it in the software that does the aerodynamic analyses, in order to have realistic values of the air flow. But this idea was quickly put aside, since it was impossible to do with the available computational resources. So, to simplify the problem in a computational view, it was decided to analyze only the wheel, with the influence of the road.

Fig. 7 – Comparison between the real wheel and the virtual one.

The control parameter that will allow us to compare the different solutions will be the mass flow rate. The more air circulation we have, the more efficient is the rim. The results obtained will not be the same as the ones that would be obtained using a real model, since the vehicle has influence in the air flow, but this approach will enlighten us to decide which shape should the rim have to dissipate as much heat as possible.

There's a parameter that must be previously defined which is the velocity of the air stream. It was defined as 80 km/h. The wheel has a diameter of 516 mm, which corresponds to a rotational speed of 87.15 rad/s.

The three-dimensional construction of the model was made in the Solid Works software, while the aerodynamic analysis was made in the Star-CCM+ software. Since the Star-CCM+ analyze the fluid flow, we need to build in Solid Works not the model of the wheel, but everything else, that represents the air. In the next picture, it can be seen a model of the wheel, and a part of its “negative”, representing the air around. The frontal face has a certain transparency, so we can see through it. In this case, the air is the material, and the wheel is void.

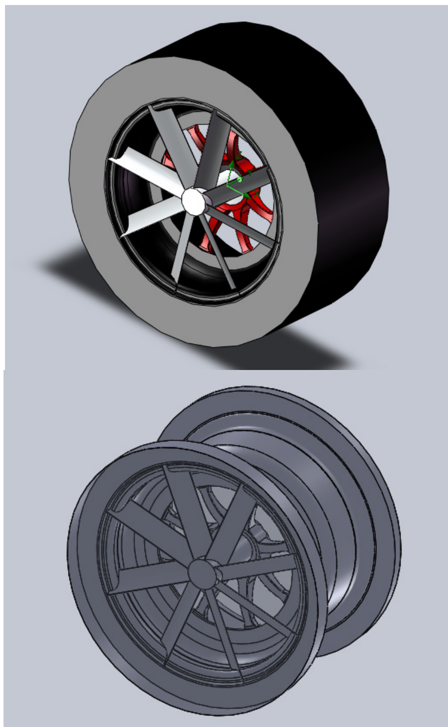

Fig. 8 – Comparison between the model of the wheel and its negative

But doing the negative isn’t enough. Since the rotation of the wheel will be analyzed, the model will have to have two distinct parts. One with static mesh, and other with a rotational mesh. In order to avoid certain problems with both softwares, the moving mesh won’t exactly match the wheel, but an internal volume that embraces the rim and a part of the tire. The moving mesh part can be seen in the previous image.

After creating the inside region, we need to model the outside, that corresponds to a large tunnel with a void where the interior region will fit. When both regions are together, we will have all the air surrounding the wheel modeled, and a void corresponding to the wheel. The tunnel needs to be large enough so the walls don’t interfere in the flow around the wheel. Its dimensions are approximately 10 times de diameter to do inlet and to the top, 20 times de diameter to the outlet and 8 times the diameter for each side:

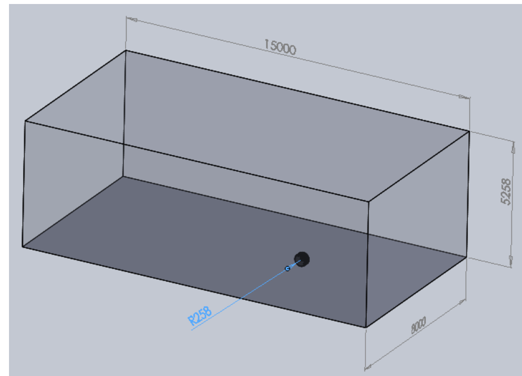

Fig. 9 – Model’s dimensions

More information about this subject can be found in (Lombard, 2009).

4. AERODYNAMIC ANALYSIS

As it was said before, the aerodynamic analyzes was made in the Star-CCM+ software. It’s one of the most advanced softwares of CFD available in the market, and the IST owns a few licenses.

Here it was imported the file created in Solid Works in the *parasolid* format, and the first steps were generating the mesh.

The choice of the mesh is apparently simple. We choose the options *surface remesher* to do a new mesh, *polyhedral mesher* is the shape of each element of the mesh and *prism layer mesher* to give more accurate information near the surfaces, to enhance the boundary layer effect.

After choosing what kind of mesh we will have, we need to parameterize it. The factor that influences the level of detail of the mesh is the *base size*, and it was given the value of 200 mm. It may seem to be a high number, but this is just a reference. Near the surfaces the mesh will be refined.

Beside the *base size*, a large number of options were also modified.

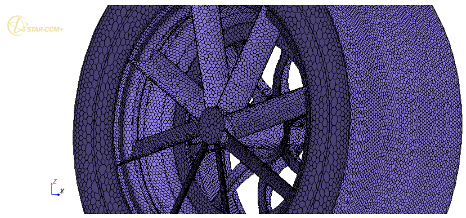

Fig. 10 – Example of a mesh

The turbulence model chosen was the *k-epsilon.* This model assumes that the flow is entirely turbulent, and the viscous effects and neglected.

In terms of the physics of the model, the simulation took 2 different steps. First, the model was made a stationary run. The mesh didn't move, but the moving surfaces had a tangential velocity. For instance, the wheel had a rotational speed of 87.15 rad/s, and the road had a velocity of 80 km/h. When the solution of this problem was found (and it was found quickly), than the mesh started to rotate. At that time, the speed of the wheel's surface was defined as 87.15 rad/s relative to the stationary referential, and the inner region containing the wheel started to move. In the beginning of this work, it was thought that the stationary analyses was enough. But in fact, it isn't.

The boundary conditions of the tunnel where the following: the velocity inlet was defined at 80 km/h, the pressure outlet is the atmospheric, the surface where the wheel touches (road) has a tangential velocity of 80 km/h, and all the other three walls of the tunnel have a slip condition, because there's no need to create a boundary layer there. The wheel has a relative rotational velocity of 87.15 rad/s.

The contact area of the road and the wheel can't be a line (it was a line if both surfaces are tangential). It's not realistic, and brings problems in the computational method. So the air near the contact area was removed, transforming the contact area in a rectangle.

Both regions have an overlapped surface, their boundaries. These two surfaces must be connected, and transformed into just one. To do that, we select both surfaces, and link them using the internal interface.

Than the time step must be chosen in the transient analyses. There are a few things the user must have in mind when choosing this parameter. We have to be sure that all the variables, mainly the air flow (Fig. 11) and the residuals, have stabilized their values before going to the next time step. But also we need to make sure that the exchange of information between cells is not compromised. It was set that each cell belonging to the rotational mesh has to interact with each cell of the stationary mesh at least four times (Fig. 12). Observing the mesh movement, it was decided that the time step should be $1.5E^{-4}$ s. To make sure there is no error propagation and every step is being done correctly, there was made a control point on a random place behind the wheel. The purpose of this point is to control the velocity in that place. This way, we can verify if the velocity is stable in the end of each time step, before going to the next one.

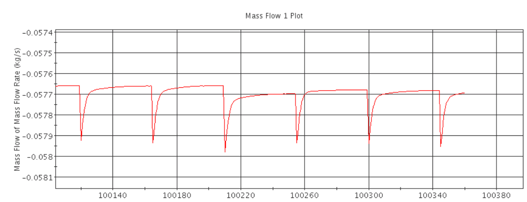

Fig. 11 – Air flow control

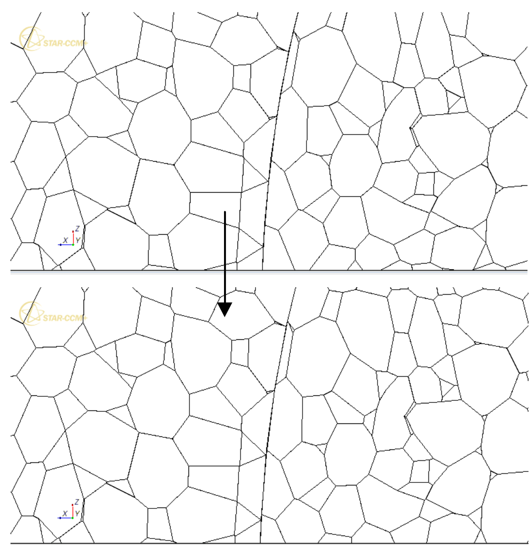

Fig. 12 – Mesh movement control

The under-relaxation factors govern the extent to which the previous solution is supplanted by the newly computed solution. By default, it is 0,7 for the velocity and 0,3 for the pressure. Since the stationary solution was having difficulties converging, this values were changed to 0,5 and 0,2 respectively. This way, the solution took more time to be found, but it did converge.

When the transient analyses starts, we need to pay close attention to the residuals. The initial iterations have high values of the residuals, so there's the need to run more iterations in each time step, or else the solution would diverge. A satisfactory value for the residual is 1E-4, but this is a very large simulation, so we just need to see when does the residual begin to be constant. The first time step take a around 200 iterations to become constant, but this number decreases very fast. In few time steps, they are only needed around 50 iterations in each time step.

More information about this subject can be found in (Anderson, 1995), (Launder & Spalding, 1979), (Patankar, 1980), (Star CCM+ User Guide) and (White, 1998)

5. RESULTS

5.1 Original Wheel

There were made three simulations. The first one consisted in the analysis of the flow around the wheel as it is (Fig. 7). It was obtained these values of air flow inside the rim for the transient run:

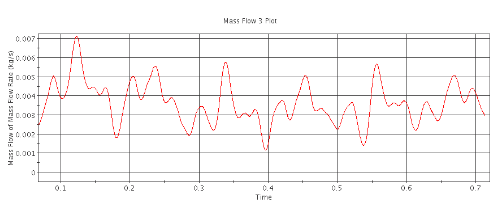

Fig. 13 – Air flow results

As it can be seen, it is very irregular. The range of values is between 0.005 kg/s and 0.002 kg/s. These values were obtained in an area of 0.8 m^2 (calculated in the software), which means that the average speed varies between 0.021 m/s and 0.053 m/s.

Despite the irregularity of these values, it is possible to observe a pattern. A full rotation is carried out in 0.072 s. If we use as a reference the lower flow peaks we verify the following pattern:

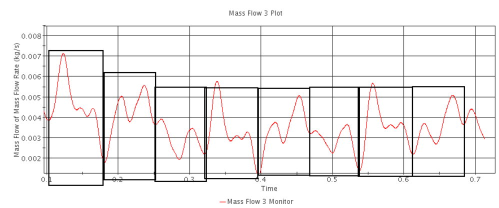

Fig. 14 – Observed pattern

The scalar view of the velocity in a plane that cuts the wheel vertically through the middle, and in a plane that cuts the wheel horizontally also through the middle can be seen in the next picture (the inlet is far from the left side):

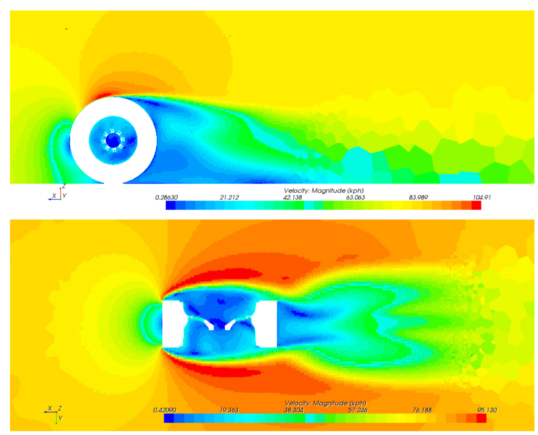

Fig. 15 – Velocity values in two planes a) vertical plan that passes in the middle of the wheel b) horizontal plan that passes in the middle of the wheel.

As it can be seen, there's a stagnation point in the front of the wheel, a low velocity volume in the rear of the wheel, which is the wake, and a high velocity volume of air in both sides of the wheel. This region of high velocity represents a problem if we want to to force the air to go through the middle of the wheel. As we can see, in this region the air moves in low velocity, and has problems to get out or in. If there were heat sources, the heat would accumulate, not helping the so needed cooling of both components. If we trace a few flow streamlines passing through some points in the interior of the rim, we have what is shown in the next picture:

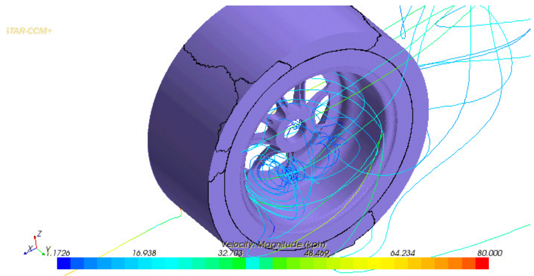

Fig. 16 – Inside streamlines

As we can see, there is too much air recirculation inside of the wheel.

Selecting a pressure distribution in the surfaces, the result is the following:

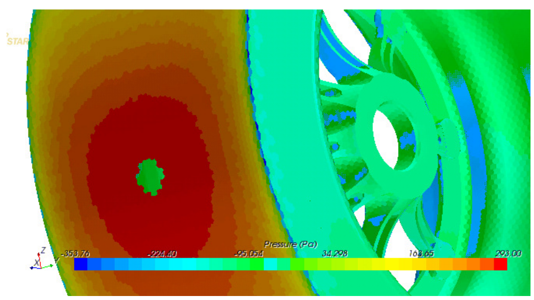

Fig. 17 – Pressure values

The maximum pressure was modified so we could find out what was the value of the stagnation pressure. It was calculated as a little bit more than 293 Pa. The maximum pressure was set in this value, and what we see is a whole in the area where the stagnation point is placed. If we had limited the pressure to 294 Pa, we would see all the area around this point in red. But this is not the point where the pressure is higher. This point is placed between the wheel and the road. Here we also have a stagnation point, plus the compression due to the wheel movement:

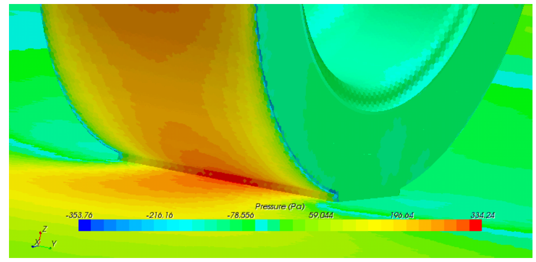

Fig. 18 – Maximum pressure (334 Pa)

5.2 Wheel with a Radial Ventilator

The second model has a radial ventilator, where the air comes from the inside region of the wheel, and leaves the wheel in the outside region in the radial direction:

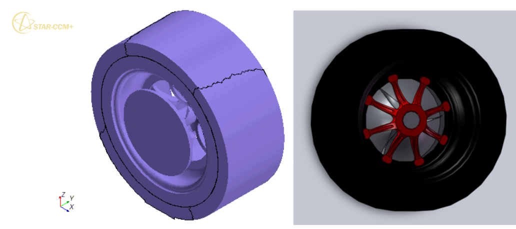

Fig. 19 – Model of a wheel with a radial ventilator

This was the first attempt to solve this problem. It has a very simple geometry with 3 small blades. The air flow results were the following:

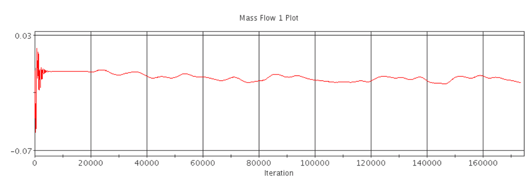

Fig. 20 – Air flow results

Here it can be seen both runs, the stationary that took place until iteration number 20000. After that, it was made the transient analysis.

After a sufficient number of iterations the flow rate varies between 0,005 kg/s 0,012 kg/s, which represents a significant increase. The area where the flow rate was measured is 0.0615 m^2, which means that the average speed varies between 0,069 m/s and 0,165 m/s

The Fig. 21 can show the air flow chart with higher definition, and the cycle of the air flow along each rotation highlighted. In this simulation it was only possible to catch two clear rotations. It should be given special attention to the presence of three air flow peaks corresponding to the passage of each of the three blades by the upper vertical position in each rotation.

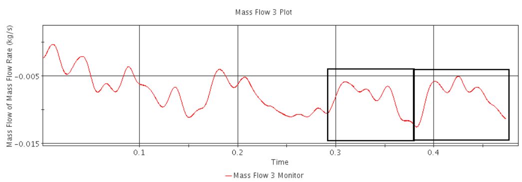

Fig. 21 – Detailed air flow results.

This represents a small improvement in the air flow. With this solution the air doesn't stack up so much as it did before, which means more cooling capacity, as it can be seen in the next picture:

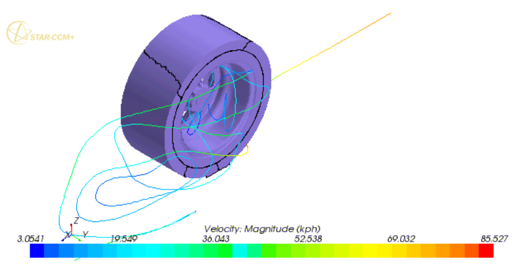

Fig. 22 – Streamlines passing through the rim. The inlet is on the right side.

5.3 Wheel with an Axial Ventilator

The third model has an axial ventilator, where the air comes from the inside region of the wheel, and leaves the wheel in the outside region in the axial direction:

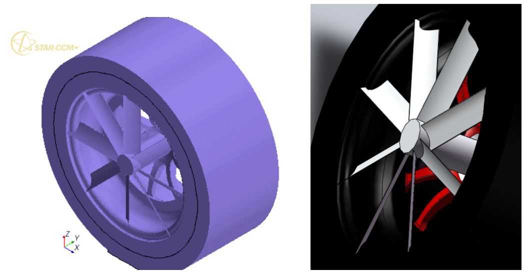

Fig. 23 – Model of a wheel with an axial ventilator.

This was a more complete ventilator, with 8 larger blades. The air flow results are shown in the next picture:

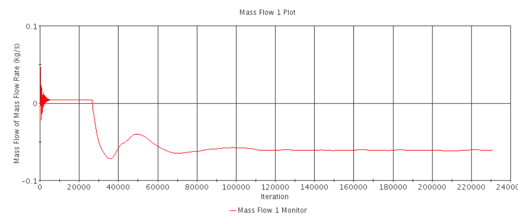

Fig. 24 – Air flow results.

Here it can be seen both runs, the stationary that took place almost until iteration number 30000. After that, it was made the transient analysis. As we can see, the results obtain in each run are completely different. The air even flows in different directions. The stationary solution is much faster to find than the transient one, but it is not accurate. Watching the Fig. 23 we can see that the air velocity must have a negative value. Taking a closer look to the stable region of the transient air flow results, and after figuring out the rotating cycles, we obtain what can be seen in the next picture:

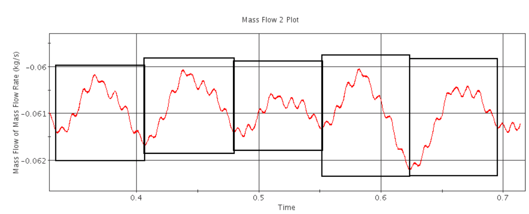

Fig. 25 – Detailed air flow results.

There can be distinguished five complete rotations, where the flow shows the same pattern. The air flow values vary between 0.06 kg/s and 0,062 kg/s, which not only represents a vast improvement in the air flow, as also demonstrates a great stability of the flow. In each rotation we can see eight flow peaks corresponding to the passage of each blade by the upper vertical position. The area where the flow analysis took place is 0,078 m2, and the average speeds vary between 0,649 m/s and 0.67 m/s.

In the next picture we can see the evolution of the speed registered in the control point. We can see the stationary run, which converged relatively fast, until iteration number 40000. After iteration number 120000 the transient run apparently stabilized in this point.

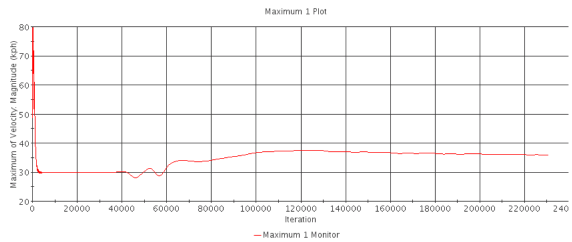

Fig. 26 – Velocity in the control point

Looking closely to the horizontal center plan of the wheel, we can see the following vectorial distribution of the velocity:

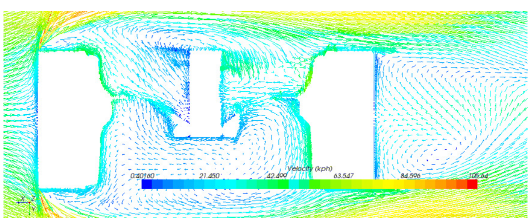

Fig. 27 – Velocity within the wheel.

We can see in Fig. 27 that the rim still have problems draining the air from the inside. Near the ventilator the air should go out throughout this region, and what we see is the air going out in the front of the wheel, and part comes back to the inside. Also we can see the difficulty that the air has entering the wheel (low zone of the same figure).

The streamlines passing through the middle of the wheel are the ones in the next picture:

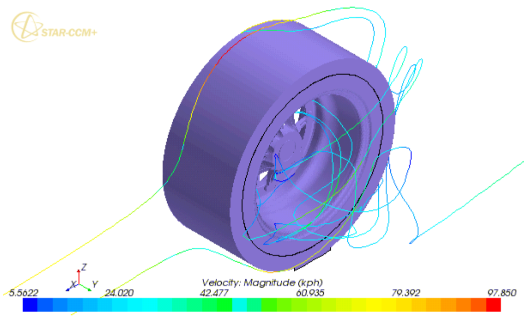

Fig. 28 – Streamlines passing through the rim. The inlet is on the left side.

We can see that more air is coming in and going out, which means (also looking to the other results) that this solution is much better than the other

6. CONCLUSIONS

Of the few tests carried out, there is a big improvement in the air flow through the rim with 8 axial blades. The results are summarized in Table 1.

As we can see, the last solution brought great improvements to the flow, although it’s still not enough. We can see a large volume of air that has difficulties getting out of the rim.

One of the advantages of the radial ventilator is that it can be added a system with bearings in the supportive axis, as what exists on bicycles, that on the occasion of a deceleration, allows the ventilator to continue spinning with the speed of the wheel at the time immediately before the deceleration. It would be a relevant improvement, since it is precisely when the vehicle brakes that is generated more heat in this area, due to heat dissipation in the brakes. This system cannot be adapted in the axial ventilator, since the blades are unified with the rim.

Table 1 – Results

	Original Wheel	Radial Ventilator	Axial Ventilator
Maximum air flow [kg/s]	0.002	0.005	0.06
Minimum air flow [kg/s]	0.005	0.012	0.062
Mean velocity in the control area [m/s] (corresponding to the minimum air flow)	0.021	0.069	0.649
Mean velocity in the control area [m/s] (corresponding to the maximum air flow)	0.053	0.165	0.67

7. REFERENCES

Anderson, J. D. (1995). *Computational Fluid Dynamics, the basics with applications.* McGraw-Hill International Editions..

Launder, B. E., & Spalding, D. B. (1979). *Lectures in mathematical models of turbulence.* Universidade de Michigan: Academic Press.

Lombard, M. (2009). *SolidWorks 2009 Bible.* Wiley.

Patankar, S. V. (1980). *Numerical heat transfer and fluid flow.* Taylor & Francis, Inc.

Star CCM+ User Guide. (n.d.).

White, F. M. (1998). *Fluid Mechanics.* Rhode Island: McGraw-Hill.