IMPLEMENTING ADVANCED CAE TOOLS IN AUTOMOTIVE ENGINEERING EDUCATION AT CHALMERS UNIVERSITY OF TECHNOLOGY

Lasse Christoffersen*, Christoffer Landström, Lennart Löfdahl
Chalmers University of Technology/Applied Mechanics, Sweden

KEYWORDS –
CFD, Course, CDIO, Aerodynamics, Formula Student, Eco-marathon

ABSTRACT –
Introducing modern CAE software in the classical engineering education is a delicate task. It is always a balance of making sure the student understand the underlying physics and working method of the software as well as giving them the skills so operate it. In this paper it is described how modern Computational Fluid Dynamics software was successfully introduced into the engineering education at Chalmers University of Technology. In the paper the outcome of some of the simulations performed in the course as well as the outcome of two case studies is presented.

The outcome of introducing modern CFD software in the education is an increased job market value for the students and several of the course participants have gone straight into CFD related positions in industry.

TECHNICAL PAPER -

1. INTRODUCTION

Introducing CAE software in engineering education is a balance between securing an understanding for the underlying physics and numerics of the simulation approach and giving the students relevant working knowledge of the latest software. Historically there has in academia been a strong focus on the former, with students writing simulation code in various programming tools. While this is good from an educational point of view by providing an insight into the physics and numerics, it does not introduce the students to recent software technologies used in industry. In the past, this was not an important concern since the gap between the two approaches was insignificant. However, in recent years the gap has increased and therefore it is believed at Chalmers University of Technology that to provide the students with the best possible skills and get them ready for industry or advanced research they should be introduced to the latest CAE technologies during their education.

In the Advanced Road Vehicle Aerodynamics course given at Chalmers, the students are studying the aerodynamics on a semi detailed road vehicle with the use of Computational Fluid Dynamics based on an industrial approach. In the course ANSA from Beta CAE constitutes on of the key CAE software used.

This paper describes how advanced CAE tools can be introduced to undergraduate students with high success. The paper will go through the simulation strategy adopted in the course and give examples of the outcome. Furthermore it will be presented how the CAE tools later are applied in various “Design and Build” project courses like the Eco-Marathon and Formula Student/SAE at Chalmers and the result of this.

2. ADVANCED COURSE IN ROAD VEHICLE AERODYNAMICS

The aim of the advanced course in Road Vehicle Aerodynamics at Chalmers is twofold. First it introduces the students to advanced, commercial Computation Fluid Dynamics (CFD) tools and provides the students with an understanding of the industrial approach to aerodynamic CFD simulation. Second it focus in detail on some of the more advanced fluid dynamic phenomena that are developed on a modern road vehicle. Every year only a small number of students, ranging from 8 to 10 are allowed to enter the course due to limited hardware resources. Since the aim is to do realistic industrial simulations it requires adequate hardware to do so and such hardware has a significant cost associated.

Learning Approach

During the course there is a great emphasis on "learning by doing" and hence the students spend a great number of hours on familiarizing and learning the course software. Furthermore, there are a number of lectures that introduce the software and explain the theory and working principles behind them as well as lectures that cover the relevant fluid- and aerodynamic phenomena. In addition there are usually two to three guest lectures where the lectures are professional aerodynamicists from industry.

The geometrical models used in the course have been semi detailed models of passenger cars that have been obtained through collaborative partners as SAAB Automobile and Volvo Cars. Lately the SAAB 9-3 convertible and the Volvo VRAK have been the preferred models. Of the former Chalmers has a 1:5 wind tunnel model which enables the students to compare their CFD simulations to actual wind tunnel measurements.

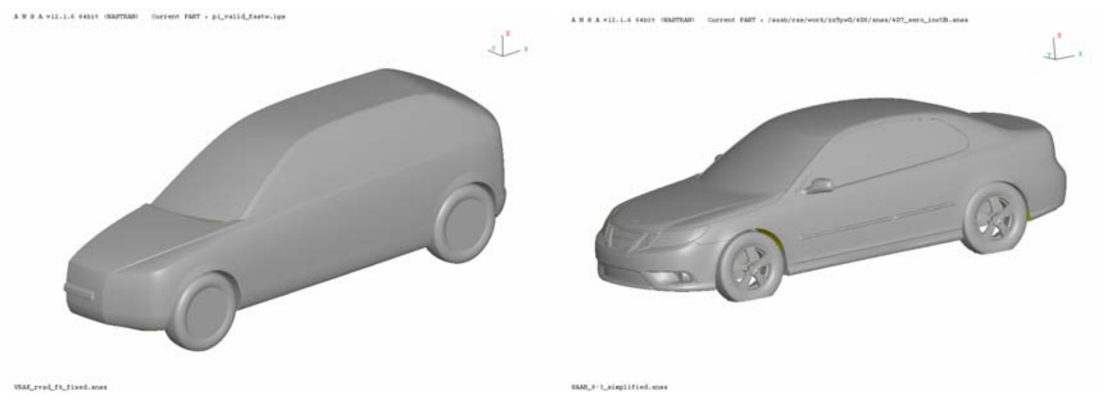


Figure 1 – The SAAB 9-3 and Volvo VRAK models used in the Advanced Road Vehicle Aerodynamics course at Chalmers.

Course Software

Although one aim of the course is to introduce the students to advanced, commercial CFD tools it is not intended to make them expert users of the software chosen for the course. Therefore software that enables the students to get to the analyzing stage faster and provide an accurate solution has been chosen. ANSA from BetaCAE is the preferred CAD clean-up and surface meshing tool. Harpoon from Sharc LTD is used for volume meshing and Fluent is used as solver and as post-processor.

As ever with CFD simulations a large amount of the time is spend pre-processing the geometry for the simulations. However, by choosing relative simple models it is possible for the students to spend up to 50% of their time on analysing the results and gain a better understanding of the physics.

Course Simulation Results

Below a sample on some of the simulations results that has been achieved though the course can be found. The simulations of which the results are presented have all been of the Volvo VRAK. The volume mesh size of each case was in the region of 3-4e+06 cells. Fluent 6.3 was used as solver and 2nd order discretization schemes were used.

Figure 2 shows surface pressure coefficients for the front part of the VRAK vehicle. By studying basic quantities such as this the students get a good overview of the flow field around the vehicle. Typical areas such as stagnation pressures in the front and the accelerated flow over the a-pillar can be identified. To furthermore visualize this it is common that the students use path lines to track the flow. Figure 3 and 4 show some typical path lines visualizing some of the stronger vortices around the VRAK model. Another typical example found in textbooks is centreline pressure distribution for passenger cars. Figure 5 show centre line pressure coefficient for the VRAK model. By plotting such properties the students are able to connect the literature to actual results which is believed to increase the learning outcomes.

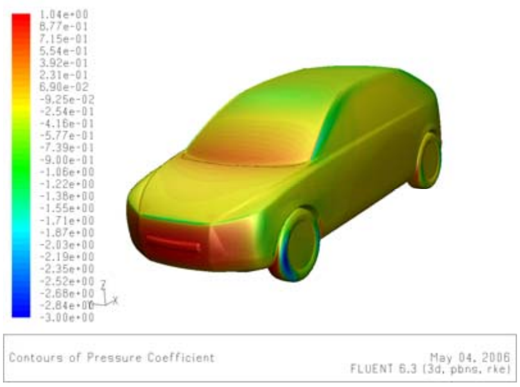


Figure 2 – Surface pressure coefficients on the VRAK.

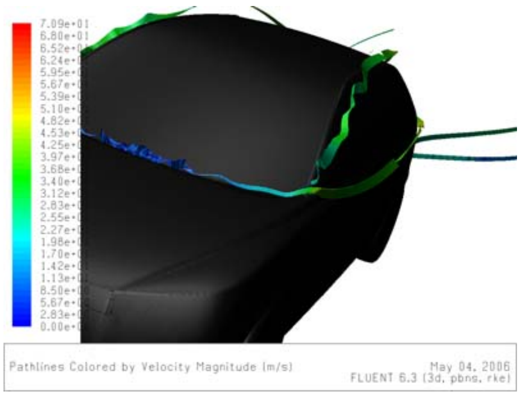


Figure 3 – Streamlines attempting to capture a-pillar vortex structure.

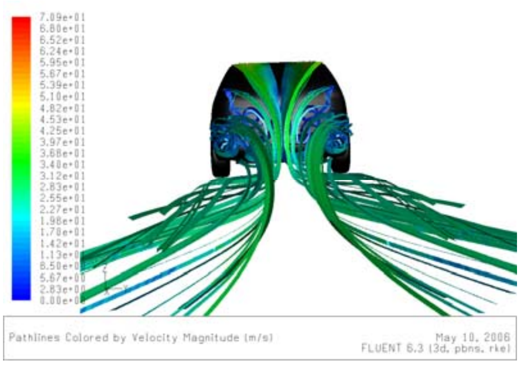


Figure 4 – Streamlines in the wake of the VRAK model.

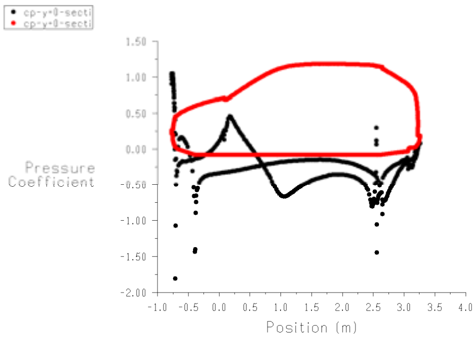


Figure 5 – Centreline pressure coefficient for the VRAK.

3. USING CAE IN PROJECT COURSES AT CHALMERS

Once the students have completed the Advanced Road Vehicle Aerodynamics course at Chalmers they are encouraged to utilize their knowledge and expertise in some of the project courses that are offered at the department of Applied Mechanics at Chalmers. Chalmers has through a number of years had two large project courses on the curricular: Formula Student and Ecomarathon. Common for both courses is that a vehicle has to be designed and build during a school year. In the Formula Student project the product created should be a small formula racecar for auto cross events and in the Eco-Marathon project the goal is to bring forward a vehicle that can go as far as possible on a specific amount of energy. At the end the particular vehicle is entered in either the British Formula Student competition or the Shell Eco-marathon respectively. The two project courses are conducted along the guidelines of the international educational concept: CDIO [1].

CDIO

CDIO is the abbreviation for Conceive, Design, Implement and Operate and is an educational concept that has been around for some years now. The idea is that through a design, build and test project, the students should be able to link the theory they have learned during their studies and create a real world mechanical system. Of course it doesn't have to be a mechanical system, it can be an electric or actually any kind of system. The vision is that this will create a new breed of newly graduates that can adapt to the industry more rapidly.

Normally the duration of a CDIO project course will be split into four parts were each part or phase representing one of the letters of the CDIO concept. For the course the intention is that the students should go through each phase.

In the conceive phase the students should define the concept of their creation and pin point the strength and weaknesses of this. In addition they should form the sub-groups needed as well as make detailed time tables etc. Through the design phase the system will be designed in details and most often this is where the students can apply the more tough theories they have learned. In the implementation phase the students will spend most of their time in the workshop building their system. For the aforementioned projects this phase is conducted in the Prototype workshop found in the Applied Mechanics building at Chalmers. This phase is usually the one providing the biggest "aha" experiences, meaning that the students discover that theory doesn't always correlate with reality.

In the implementation phase it is time to test the system. This is probably where the biggest reward for the students is awaiting. It is fantastic to test your own creation and find that it is actually working as intended.

At Chalmers the project courses are offered for the students on their final year of studies. In this way they will experience real project work and hopefully boost their confidence before they go into industry.

To show how modern CAE tools are used in the CDIO project courses at Chalmers two case studies are presented below.

4. CASE STUDY 1: THE 2007 CHALMERS FORMULA STUDENT CAR

Chalmers University of Technology developed and produced a new car for the 2007 Formula Student competition, which is the British equivalent to the Formula SAE series. During testing it became obvious that the cooling capacity of the car was inadequate as overheating was experienced at several occasions. Therefore, the team wanted to find the reasons and solutions to these overheating problems. Since cooling systems relies on convective heat transfer, securing an appropriate flow of air to the radiator is fundamental to a good heat transfer rate [2]. A first step in the process of solving the problem of overheating was to improve the aerodynamic shape of the racecar in the vicinity of the cooling air inlets. As detailed CAD-data of the car was available it was decided to execute this work by the use of Computational Fluid Dynamics (CFD). Since the bodywork already had been built it was crucial that the necessary modifications did not involve major changes to the vehicle body.

The racecar is an open wheeled formula design built according to the Formula SAE 2007 regulations [3]. It is based on a tubular steel frame where the bodywork is not an integral part of the structure of the car. However, the team desired a retro-look the bodywork that was styled along the theme of the Lotus Type 49 from 1967 as seen in figure 6.

Figure 6 – The 2997 Chalmers formula student car and on the right a close-up on the right sidepod where the radiator is located.

The cooling package of the car consists of only one radiator with an electric fan mounted to the back. The package is located in the right sidepod. In the initial design the radiator was placed in the sidepod without any ducting.

Case Definition and Setup

For the study four base geometries were specified and used in cases designated 1, 2 ,3 and 4 respectively. In the cases the ducting geometry for the radiator was altered in steps. For the first case was the ducting configuration identical to the original vehicle. For the second case was the seal between the right sidepod wall and the radiator removed. The last two cases had an entirely new duct design.

To make the simulation and optimization as realistic as possible the fully detailed CAD geometry of the car was used. The geometry was prepared using ANSA from BetaCAE and from this the volume mesh was created using TGrid. To accurately capture surface forces prismatic layers were used on the body of the car. The complete hexahedral core mesh contained around 40 million cells and a section of it is illustrated in figure 7.

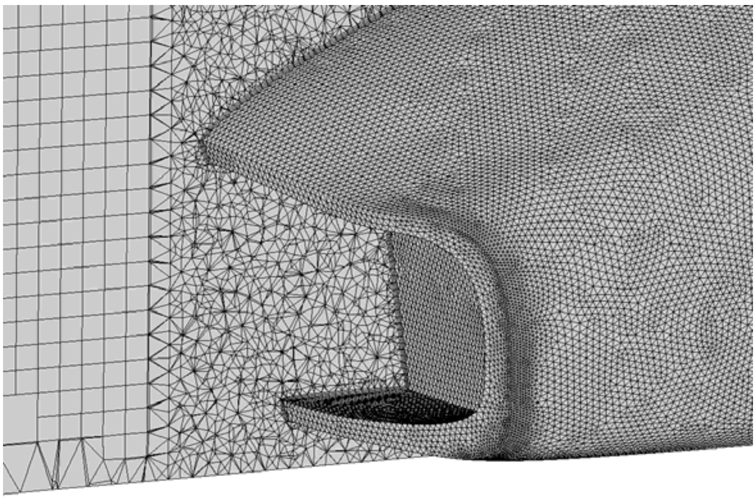


Figure 7 – The mesh at the front section of the car.

The cases were solved using Fluent 6.3.26 and the realizable k-epsilon model was used with second order discretization scheme for momentum, kinetic energy and dissipation rate. Although the k-epsilon model has several drawbacks in terms of physically modeling it has proven to give fairly accurate results in road vehicle aerodynamics applications with good stability and was therefore chosen [4].

The radiator was modeled as a *porous media fluid zone* which is basically a sink term added to the governing equations. All simulations were run isothermal and as a consequence the heat exchange from the radiator to the surrounding fluid was not predicted. In this study it was only pure aerodynamic effects that were of interest. For the simulations of the car the velocity was set to 80 kph which is around the average the race car achieves in a straight line.

A virtual wind tunnel was used for the simulation. This had a frontal inlet area of 60 m2 yielding a blockage ratio of 1.3%. The length of the tunnel was 50 m and the car was placed 4 car lengths behind the inlet which should be sufficiently far away to limit the interference effects between the car. Velocity inlet condition was used as inlet and symmetry condition was used for the side walls except the floor which, as previously mentioned was set with a translation velocity as the velocity inlet. Finally a pressure outlet condition was used for the exit of the tunnel.

Results of Formula Simulations

The improvement that was made in cooling air mass flow rate can be seen from the graph in figure 8. As seen with a proper designed cooling duct installed a significant gain can be achieved. In this case the cooling air mass flow rate went up with 206% and the aerodynamic drag was kept at the same level.

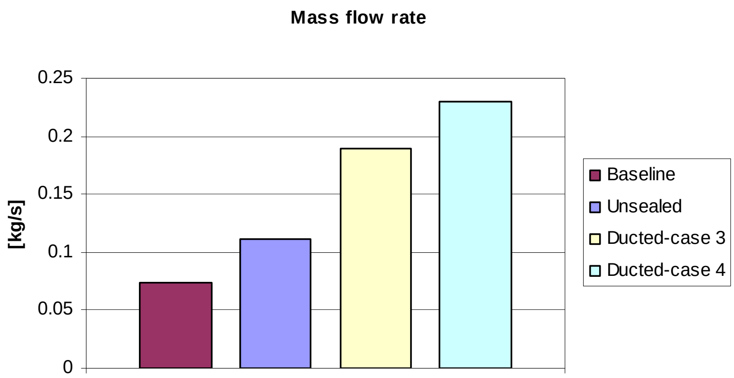
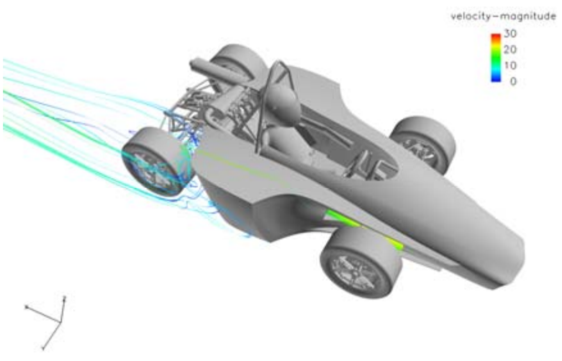


Figure 8 – The mass flow rate of the four cases.

To illustrate the improved cooling flow behavior with the duct of case 4, streamlines released in front of the sidepod can be seen in figure 9.



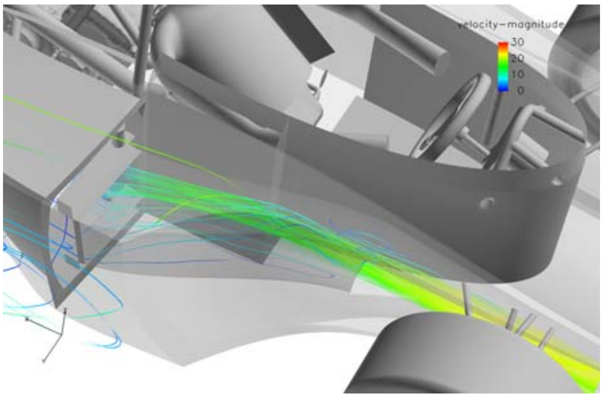


Figure 9 – Streamlines illustrating the improved cooling flow.

For more details on this case study please refer to Christoffersen et al. [5].

6. CASE STUDY 2, THE CHALMERS ECO-MARATHON VEHICLE "VERA"

Designing a vehicle to go as far as possible on a certain amount of energy is done by reducing the driving resistance as much as possible. The aerodynamic drag of an eco-marathon vehicle is of extreme importance. Therefore detailed simulations on the Chalmers Eco-marathon vehicle "Vera" was carried out to access and improve the vehicle. Vera competes in the prototype class in the international Shell Eco-marathon series and currently holds the Swedish record of 1243 kilometres on a litre of gasoline. In figure 10 Vera can be seen.

Figure 10 – The Chalmers Eco-marathon vehicle Vera.

Since it is a very streamlined vehicle a good quality mesh is needed to make a trustworthy simulation. Resolving the boundary layer developed on the body is essential to predict separations. To do so, the body of the vehicle was meshed with 10 prism layers with the first node placed at $y^+ = 1$. To create the surface mesh that allowed such a good creation of prism layers ANSA was used. The volume mesh was created with Tgrid from Ansys. An illustration of the surface mesh can be seen in figure 11.

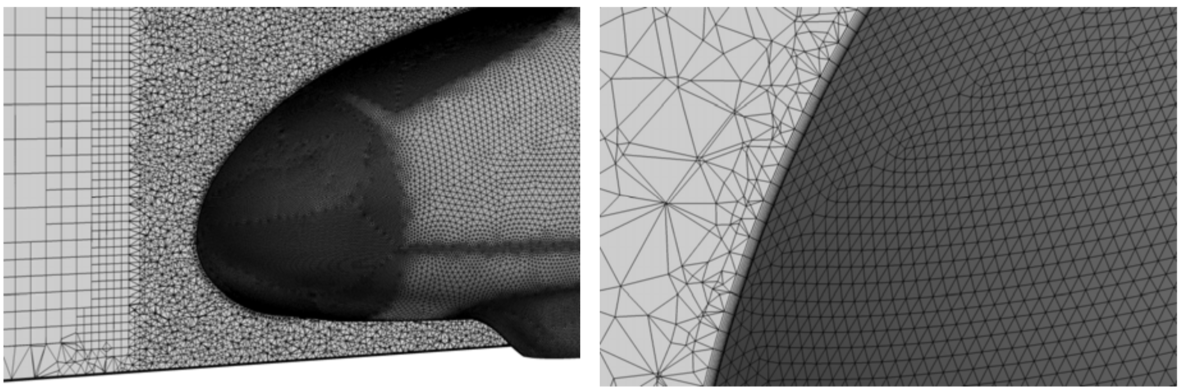


Figure 11 – A cut-out of the mesh. On the right a zoom-in image displays the 10 layers of prism cells.

To see how Reynolds number sensitive the vehicle is a Reynolds number sweep was conducted with a free stream velocity ranging between 25 to 35 kph. Due to the low average speed that is dictated by the regulations sidewind becomes a possible concern in the sense that drag can increase substantially if not careful considerations in the design phase are made. Therefore a yaw sweep from 0 to 25 degrees was also conducted. The yaw sweep was only done at a vehicle velocity of 30 kph. Furthermore by knowing both the sensitivity to Reynolds number and sidewind it is possible to improve the driving strategy by taking the local wind conditions at the track into account.

In the baseline configurations at a free stream velocity of 30 kph Vera has a coefficient of drag of 0.108 and a coefficient of lift of -0.131. In figure 12 the coefficient of pressure on Vera can be seen and in figure 13 parts of the flow field is displayed with streamlines released 0.6 meters in front of the vehicle. From figure 13 it can be seen that the parts of the vehicle that cause the greatest disturbance to the flow field are the openings in the wheel spats. Two vortices are created at each wheel spat. For the front wheel spats the vortex on the outside is significantly stronger than the inside one and is trailing a considerable distance downstream. Despite of these unfavourable vortices the majority of the drag is caused by viscous effects which is a common phenomena for streamlined vehicles like Vera.

Figure 12 – The coefficient of pressure on Vera.

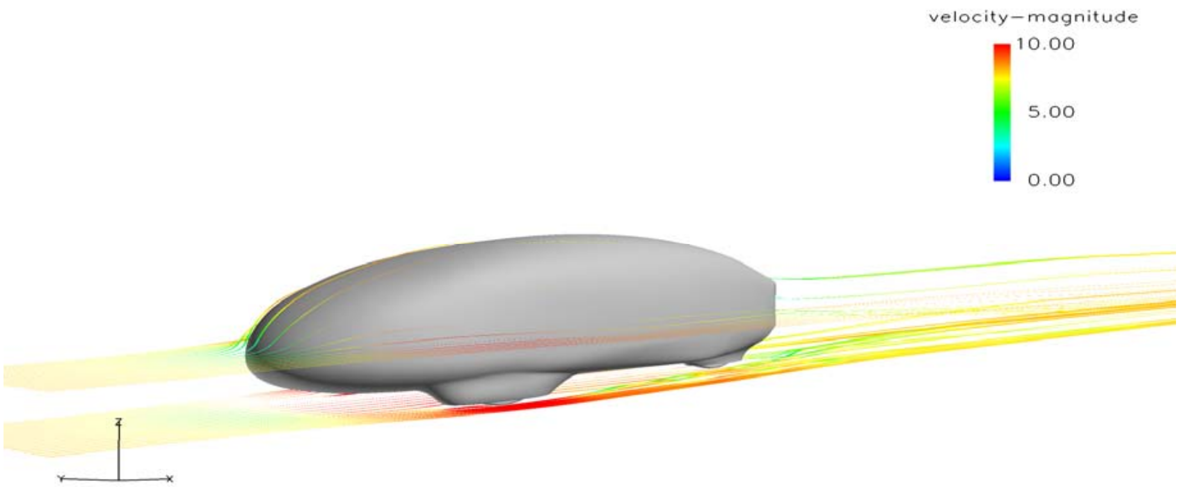


Figure 13 – Streamlines visualizing the flow field around Vera.

7. OUTCOME OF THE CAE EFFORT

Although there from an academic point of view always is an interest in teaching young minds there is a requirement from the industry that what is taught enables the students to fill an employment position without the need of too much additional training. Having provided the students with the skills to conduct aerodynamic CFD simulations with a modern industrial approach have been greatly appreciated by the industry in general. Of the students who have made it through the Advanced Vehicle Aerodynamics course a large number have gone directly into CFD related jobs in, as an example companies like Red Bull Racing, Volvo Cars, SAAB Automobile, Volvo AB etc.

8. SUMMARY

In this paper it is described how advanced computational fluid mechanics tools have been introduced in the Advanced Road Vehicle Aerodynamics course at Chalmers. The course software line-up as well as examples on the outcome of a number of simulations was presented.

Two case studies of aerodynamic simulations performed in CDIO project courses at Chalmers was also presented. The first being of the 2007 Chalmers Formula Student car, where the cooling air mass flow rate was improved with the use of CFD. The second case study presented how the aerodynamics of the Chalmers eco-marathon vehicle Vera was studied using CFD. In both cases students have been using the knowledge and skills they obtained through the Advanced Road Vehicle Aerodynamics course to perform the studies.

For the students who have been involved the in the Advanced Road Vehicle Aerodynamics course at Chalmers the transition into CFD related positions in industry has been smooth. Today students of the course are presented in a range of international companies. Furthermore, the course have been greatly appreciated by the industry in general.

REFERENCES

(1) www.cdio.org, October 2006.

(2) Incropera, F. P. “Introduction to heat transfer” 5th edition, Wiley 2007.

(3) 2007 Formula SAE Rules, Society of Automotive Engineers, 2006.

(4) FLUENT 6.3 User’s Guide, Fluent Inc. 2006-09-20.

(5) Christoffersen, L. M. ; Söderblom, D. ; Löfdahl, L. (2008). Improving the Cooling Airflow of an Open Wheeled Race Car. Motorsports Engineering Conference & Exposition. (SAE 2008-01-2995)