On the thermodynamic development of the New Volvo XC 90 using FLUENT.

Anders Jönson
Volvo Car Corporation,
Sweden

ABSTRACT

In order to meet changing consumer demands, car manufacturers are offering the customer more diverse products these days. The new XC90 represents the first SUV to be offered by the Volvo Car Corporation, the development of which created new thermodynamic challenges. Increased vehicle weight, heavier trailer loads, and higher aerodynamic drag would require up to 50% more cooling airflow to the cooling package than the recently launched S80 saloon. Ground clearance regulations and large frontal area also yielded high demands on front end optimisation.

This paper describes how some of those challenges were approached and investigated using FLUENT. Areas of specific interest include front end design, cooling package layout, bumper beam shape and position etc, meeting the increased cooling requirements of an SUV.

The paper also describes the CFD method development during the project. FLUENT was used to couple the cooling air flow side to the hot side of the heat exchangers through the use of User Defined Function programming. Comparisons with experiments verified the underlying theory and showed good accuracy of the approach and this method is now currently used in all projects where cooling performance is addressed.

Keywords: Air intake, Design Process, Volvo XC90, Sports Utility Vehicle, aerodynamics, ANSA, Fan modeling, CFD validation

1. INTRODUCTION

1.1 Commercial Background

Increasing competition within the automotive industry has led to a rapid expansion of the different types of vehicle offered to the customer. One segment of the market that has attracted more and more customers during the last decade has been that of the Sports Utility Vehicle (SUV). The popularity of this type of vehicle originated in the United States but has since become increasingly popular within Europe. Initially these vehicles were often based on an existing 4WD truck chassis, and thereby inherited similar ride and handling qualities as well as high aerodynamic drag. As the demand has become greater however, new variants on this theme have become popular. In 1999 the Volvo Car Corporation decided to enter the market using not truck-based chassis, but a saloon car platform as a base. This would offer saloon car levels of ride comfort and crash safety combined with the visibility and interior space of an SUV.

For Volvo it was inherent that the vehicle would offer predictable handling characteristics as well as competitive fuel consumption. Equally as important, the product should be available on the market as quickly as possible, use as many carry-over components from the parent platform as possible and use existing production facilities. These decisions led to the choice of the recently launched

S80 saloon as the platform base from which to work and the manufacturing centre being in the same factory in Gothenburg, Sweden. The car was to be launched onto the market in the second half of 2002. In order to minimise development costs and reduce lead times no prototype tooling would be made, the production tooling being ordered as soon as the design was frozen. Consequently there would be no early prototype series available with which to make complete vehicle tests before ordering production tooling.

1.2 Challenges

Given the prerequisites outlined in the previous section, several aerodynamic and thermodynamic challenges for the project became clear. These properties are not independent of each other, e.g. the cooling airflow going through the heat exchangers and engine compartment will contribute to drag and lift. The aerodynamic aspects of the development are further described by Walker [1]. The main challenges can be summarised as:

- The high ground clearance together with a high seating position combined with packaging requirements for the occupants, including those occupying the 3rd row seats would give a large frontal area.

- In order for the vehicle to be classified as a Light Truck in the United States, a certain number of ground clearance conditions had to be fulfilled. These are summarised in Figure 1.

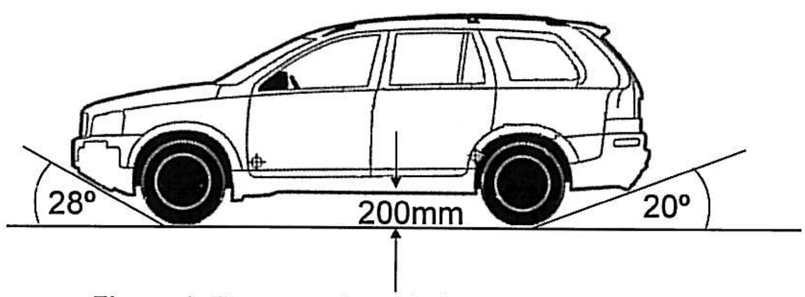

Figure 1. Ramp angle and clearance requirements

- The increased vehicle weight, heavier trailer loads, and higher aerodynamic drag would require up to 50% more cooling airflow to the cooling package.

This, together with the demand for short lead times and the deletion of the early prototype series, made CFD an important tool for the thermodynamic development of the car.

1.3 Thermodynamic Demands

There are many areas that are effected by the thermodynamic situation, e.g. cooling performance, components in the engine compartment, brake cooling etc. To ensure full functionality for the customer, specific targets were defined for systems and components. These targets have to be met by the final production vehicle. The main thermodynamic target for the cooling package is the radiator top water temperature. For continuous drive conditions it must not exceed 115 °C and for peak loads the maximum temperature is 125 °C. The fulfilment of the target is verified with measurements of a vehicle with production status.

In the development of the S80 car, FLUENT was used for cooling airflow development. An in-house computation procedure had been defined, stating how cooling air flow simulations should be performed in terms of necessary geometric detailing, mesh sizes, turbulence and discretisation models etc. One benefit of a computational procedure, also called a CFD method, is that it ensures to some extent that the computation and the result will be independent of whoever performs it. Thus, an existing CFD method could be offered to the XC90 project where different geometric designs and cooling package layouts could be evaluated in terms of cooling air mass flow. The cooling air mass flow is not only interesting for the radiator top water temperature, it also sets the temperature level of the air entering the engine compartment, thus being an important parameter for component temperature together with radiation and conduction.

It was decided to use CFD in the XC90 project and focus on the cooling air mass flow as evaluation parameter. The target was to increase the cooling air mass flow by 50 % when compared to the Volvo S80.

2. OVERVIEW OF THE PROJECT

A time overview spanning the Volvo XC90 project is shown below in Figure 2.

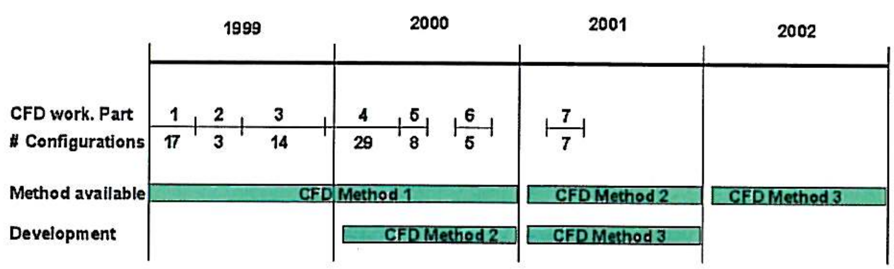

Figure 2. Time overview spanning the XC90 project

As can be seen, the majority of the CFD simulations, or configurations, where done during the first half of the project. The main reason for this is that in the beginning of a project very few parameters are fixed. Several different designers work on different proposals for the exterior, the cooling package layout is not decided and the crash structure with bumper beam positions is not decided. Since all of these areas affect the cooling air flow situation, the need for fast answers and short loop lead time led to extensive use of CFD early in the project. Examples of how FLUENT was used in the different parts are given in section 3. Figure 2 also shows at what time the different CFD methods were available and when new methods were developed. An overview of the solution strategy for the different methods is shown in the appendix and the driving forces for new development is discussed further in section 4.

3. DEVELOPING THE VOLVO XC90 USING FLUENT

One of the major advantages with CFD as a development tool is the fast lead times. A question raised at a meeting, for example the effect on the cooling airflow when increasing the size of the bumper beam, can sometimes be answered the next day.

Since several important areas and components in the vehicle affect multiple properties at the same time, it is not always easy to find a balanced solution. For example, a large grill opening is beneficial for cooling airflow but might not fit in to the overall exterior design concept, a high thick bumper beam has good crash behaviour but might block the cooling air flow. Therefore, it is not always easy to trace how CFD has affected the final design since the production vehicle often must be a balance between all properties. The important thing, and where CFD can play an important role, is that when balancing between properties is necessary, it can be done with as much background information as possible.

The idea in the subsequent sections of this paper is to give a short overview of how FLUENT was used in the different parts of the process. It is, of course, not a complete description of the development work but serves as examples of the type of answers the project got from CFD.

3.1 CFD work. Part 1: A 2D front study of the XC90

This was the first CFD work done in the project. It was decided to perform 2D cooling air flow studies to identify which parameters in the front design played an essential role in affecting the mass flow of air entering the engine compartment through grill and spoiler openings. It also served as a prestudy for the following 3D front study, described in section 3.2. In total, 17 different configurations were compared to a S80 reference computation. Figure 3 shows two of the configurations together with the S80 reference case.

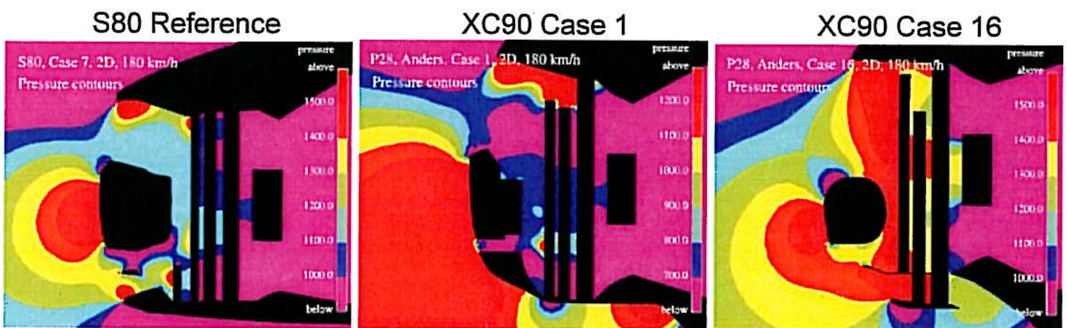

Figure 3. Illustration of the geometry and static pressure contours.

Case 1 was the first proposal, with a very small spoiler opening. The cooling air mass flow through the radiator decreased by 27 % when compared to the S80. Cases 2-15 investigated parameters such as grill and spoiler opening height, cooling package layout, bumper beam shape and size etc. Case 16 showed best performance, having 34 % higher cooling air mass flow than S80. This was mainly due to the use of an evacuation duct, discharging air underneath the engine compartment, and the smooth shape of the bumper beam.

3.2 CFD work. Part 2: First 3D front study of the XC90

This was the first 3D simulation in the XC90 project. Again, the S80 was used as reference case and the cooling air mass flow was compared for 3 configurations of XC90, all based on the exterior mode designed by Stefan Jansson. The front detailing of the S80 and Case3 is shown in Figure 4.

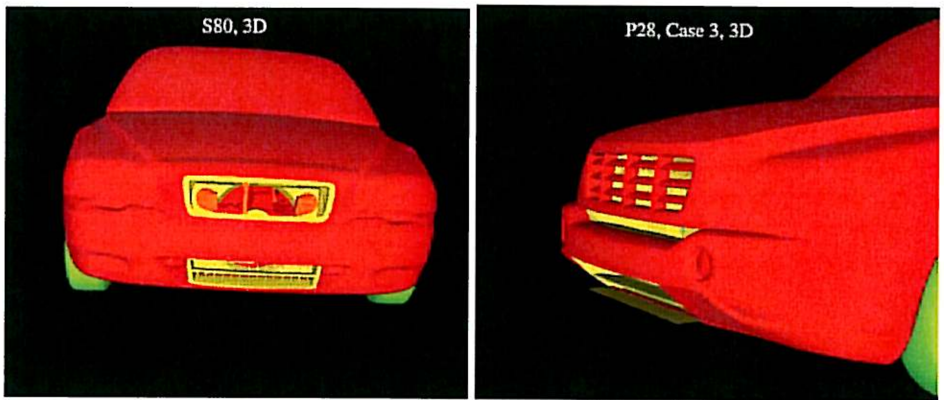

Figure 4. 3D front illustration of S80 and XC90 Case3.

All configurations, that is the S80 and the three XC90's, used the same engine compartment model. This is clearly seen in Figure 5, where cutplanes through the y=0 (symmetry plane) is shown for the S80 and XC90 case 1 and 3.

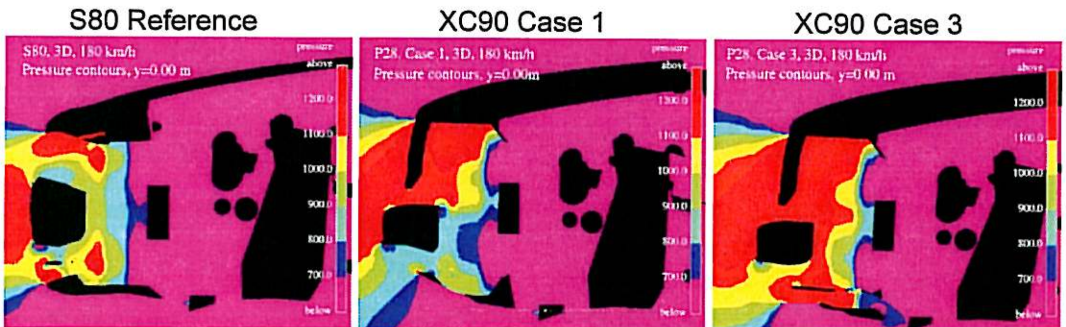

Figure 5. Y=0 cutplanes with static pressure.

Case 1 showed an increase in cooling air mass flow of 6 % compared to the S80. An increase of 33 % was reached for case 3, mainly due to a redesigned spoiler area together with an evacuation duct. This result verified some of the findings from the initial 2D study. It is also interesting to note how the restriction in front ramp angle, shown in Figure 1, has influenced the front design.

3.3 CFD work. Part 3-5

These parts span the majority of the CFD work done in the XC90 project. Main geometrical questions addressed were exterior front design, fan position, USA numberplate position, heat exchanger layout, evacuation duct and rear bumper beam shape. Non-geometrical questions addressed were fan performance, airside pressure drop of an individual heat exchanger and vehicle velocities.

As an example of heat exchanger layout, two different positions of an air-to-oil cooler was investigated, indicated by arrows in Figure 6. Case 25 has the oil cooler in front of the condenser while in case 29 it was positioned under the charge air cooler, in front of the radiator.

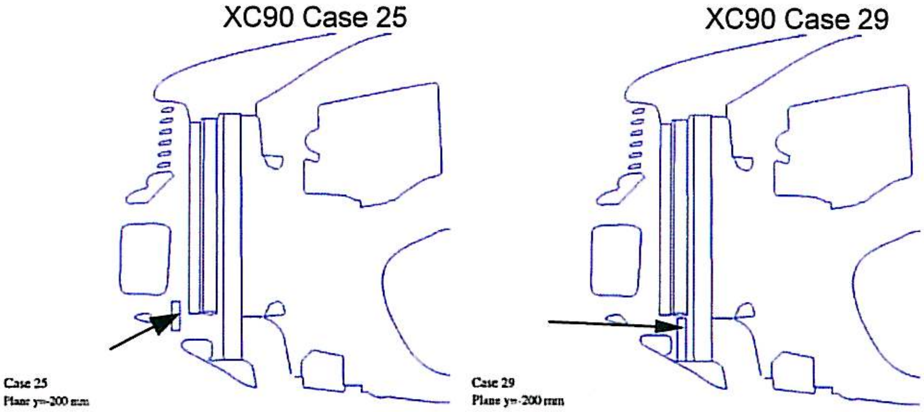

Figure 6. Illustration of air-to-oil cooler positions.

Packaging restrictions allowed an oilcooler with the width of 620 mm, height 60 mm and thickness 19 mm to be fitted in case 25 whereas case 29 used the dimensions 500x90x19 mm. The cooling airflow through the oil cooler was significantly higher in case 29, although its larger area gave a lower mean velocity over the heat exchanger. In case 25, the heat released from the oil cooler to the air will decrease the AC performance due to increased air temperature at the condenser inlet. The final recommendation to the project was to position the oil cooler below the charge air cooler as in case 29 and this is now used in production cars.

In the early phase of a project, when geometry related questions are dominating, computations are usually done only for one vehicle velocity. As the project develops, additional load cases are computed, verifying that the early solutions proposed are robust and work for all vehicle velocities. As can be seen in Figures 3 and 5, a vehicle velocity of 180 km/h was used in the earlier development work. During this part, additional velocities at idle (0 km/h) and 70 km/h were computed as a complement to 180 km/h. This led to the identification of a risk of a recirculation in idle mode for one of the configurations as can be seen in Figure 7.

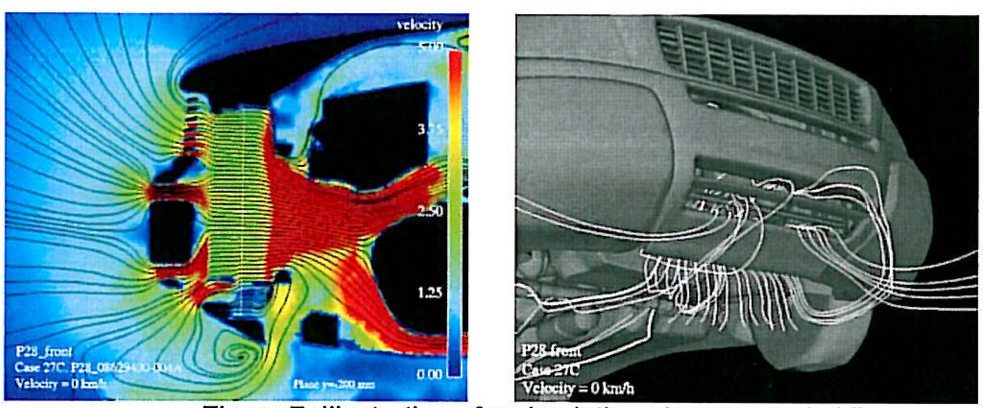

Figure 7. Illustration of recirculation phenomena in idle.

Recirculation of hot air from the engine compartment to the spoiler inlet decreases the AC performance and must be avoided. The production car has modifications of the evacuation duct and the lower crash beam, minimising the risk for recirculation of hot engine air into the spoiler opening at idle.

3.4 CFD work. Part 7

This part included a major updating of the geometry, using more or less production status for the CAD. It also included geometrical preparation of the FLUENT model for CFD method 3, described in section 4.2. However, since all other simulations were done using CFD method 1 it was decided to use that method for this part also. In that way, all results were produced using the same method and comparisons could easily be done. The final front is shown in Figure 8.

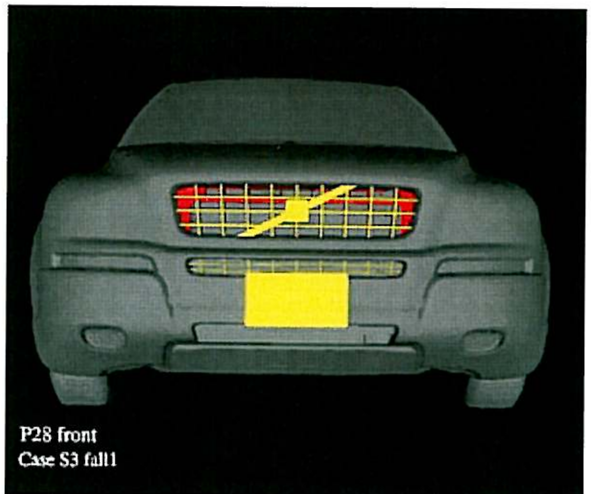

Figure 8. Front geometry used in part 7.

As can be seen, an USA numberplate is fitted to the model and the grille and openings are modelled in detail. The airflow around the complete XC90 is shown in Figure 9 together with the static pressure on the surface.

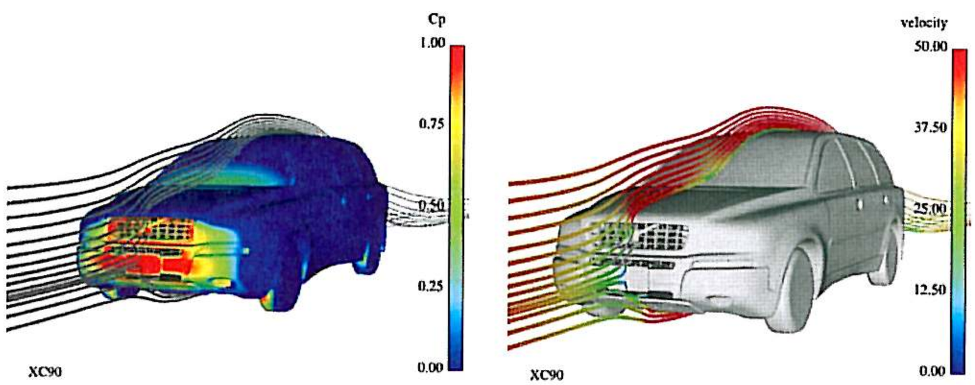

Figure 9. Streamlines around the Volvo XC90.

Finally, the cooling air mass flow for the Volvo XC90 was also shown to be approximately 55 % higher than the S80, thus meeting the targets set early in the project.

4. DEVELOPING FLUENT CFD METHODS USING THE VOLVO XC90 PROJECT

In the previous section, the development of the Volvo XC90 using FLUENT was discussed. All results and recommendations to the project were based on the parameter cooling air mass flow using CFD method 1. However, as described in section 1.3, the radiator top water temperature is the primary target for cooling performance and this led to an increasing demand of a method that could couple the cooling air mass flow to the radiator top water temperature.

Several possible routes for coupling the cooling air mass flow and distribution obtained from the FLUENT simulation to the radiator waterside were identified. One route was to use a separate heat exchanger modelling software, transferring the results from the FLUENT simulation to a format known by the separate software. This turned out to have several drawbacks; e.g. the process of translating data was not straightforward, additional software licence costs etc.

Instead, it was decided to stay within the FLUENT environment, using FLUENT's concept of User Defined Functions (UDF) to model the coupling between the cooling air and the heat exchangers. This approach offered several direct advantages; the resulting radiator top water temperature could then be obtained directly from the FLUENT simulation, in-house choice of heat exchanger theory etc. Long-term advantages such as the possibility to have the heat exchangers influencing the air flow prediction through the energy equation was also identified. The development work is described in detail in [3-7], and outlined in the following sections.

4.1 Developing CFD method 2

The increasing desire to have radiator top water temperature as the result from a cooling package CFD simulation led to the development of CFD method 2 during the year 2000. In a FLUENT simulation, the heat exchanger's core is modelled as rectangular fluid domains with specified empirical correlation for the airside pressure drop. In the UDF routines, each heat exchanger is numerically discretised and the hot side flow through the core is modelled. The overall solution strategy is briefly described in the appendix but the coupling between the airside and the coolant side is described below.

This difference between the CFD volume mesh and the numerical mesh used by the UDF routines is illustrated in Figure 10.

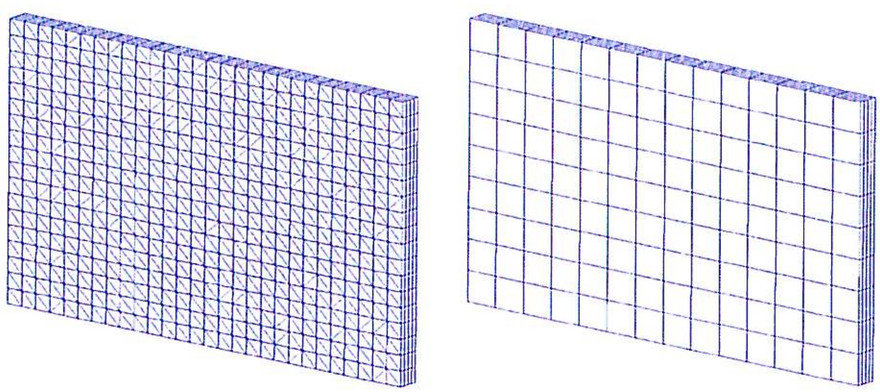

Figure 10. Left: CFD Heat exchanger mesh. Right: UDF numerical heat exchanger mesh.

A front view of a heat exchanger is shown in Figure 11.

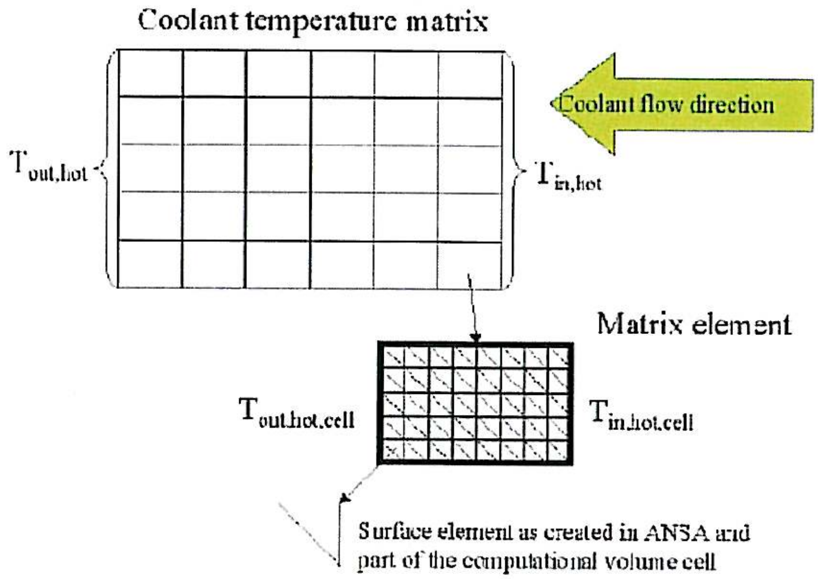

Figure 11. Schematic illustration of the meshes.

For a given heat exchanger, a specific heat rejection per area unit, q_{spec} , parameter is defined as:

$$\dot{Q}_{rad} = q_{spec} A_{rad} \left[(t_{in,hot} + t_{out,hot}) / 2 - t_{in,air} \right] \tag{1}$$

The definition of q_{spec} is assumed to be valid from a global perspective. Now, assume that q_{spec} is valid for a small portion of the heat exchanger, i.e. a cell, and thus make it possible to express the heat rejection as:

$$\dot{Q}_{cell} = q_{spec} * A_{cell} * \left[(t_{hot,in} + t_{hot,out})/2 - t_{air} \right]_{cell} \tag{2}$$

If the inlet temperature on the hot side is known, it is possible to use equation (2) and sweep along the coolant cells. Since the temperatures on the hot side need to be updated, a simple heat balance is made:

$$\dot{Q}_{cell} = \left[\dot{m}\, c_p (t_{in,cell} - t_{out,cell}) \right]_{hot} \tag{3}$$

The total heat rejection over the heat exchanger is then calculated as:

$$\dot{Q}_{total\_heat\_rejection} = \left[\dot{m}\, c_p (t_{in} - t_{out}) \right]_{hot} \tag{4}$$

In general, the total heat rejection is a given value, which means that the inlet temperature on the hot side needs to be updated to change the temperature difference in equation (2) to eventually receive a correct total heat rejection as described in equation (4).

The output of this method is the cooling air mass flow and radiator top water temperature. The limitations that can be immediately identified are that the cooling airflow is computed using constant air properties. In reality, the density and temperature of the air changes due to heat transferred from the hot side of the heat exchangers to the airside. However, these effects would also be present if a separate heat exchanger code was to be used. The developed method offered the advantages of getting the radiator top water temperature directly from FLUENT. It also enables in-house control of the heat exchanger theory used.

4.2 Development of CFD method 3

As discussed in the previous section, the limitations of CFD method 2 was immediately identified and a decision for further development during 2001 was taken. The main driving force was to improve and verify the accuracy of air mass flow rate and radiator top water temperature.

4.2.1 Coupling of the momentum and energy equations

In order to achieve accurate cooling air flow rate calculations it was identified that the energy equation had to be solved coupled to the momentum equations, using variable fluid properties. There are several reasons for this approach.

First, the airside pressure drop is dependent on the density through the heat exchanger and the airside Reynolds number. Understanding how the density varies through the heat exchanger is not straightforward but the fluid property changes are significant and must be taken care of.

Secondly, when it comes to handling the fan characteristics, a performance curve proportional to axial air velocity is used to give a discrete pressure rise/drop in the simulations. For this kind of treatment it is especially important to have accurate air velocity and not only an accurate mass flow rate, which is another crucial reason to solve the momentum equations with fluid properties which are coupled to the thermal flow field.

Finally, the pressure drop in the engine bay is proportional to the dynamic pressure, i.e. $1/2*\rho*V_x^2$, and once again, a correct air mass flow rate will not be accurate enough but correct density and air velocity is needed.

All these three phenomena evidently show the importance of the correct treatment of the density variation.

The usage of the CFD method 3 does include coupled energy equation and momentum equations through the fluid properties, in this case density and dynamic viscosity. The fluid properties will vary with temperature: density with the incompressible ideal gas law and viscosity with Sutherland's law.

4.2.2 Heat exchanger pressure drop

The pressure drop curves for individual heat exchangers should be expressed in terms of total pressure drop over the cell core obtained at adiabatic conditions. If the pressure drops are obtained at non-adiabatic conditions these must be transformed to the corresponding adiabatic pressure drops. This is to avoid counting the effect of density changes twice.

4.2.3 Fan characteristics

When it comes down to simulating the fan, the pressure rise over the blades has been found to be hard to predict; see reference [2]. One could easily blame the turbulence modeling but the geometrical representation, especially the tip clearance, is far more critical. Best practice would still be to use measured fan characteristics curves obtained with the cooling package installation. The pressure drop caused by the cooling package, i.e. heat exchanger, is then subtracted from the total measured pressure drop. In a CFD simulation, the flow in the fan shroud does create significant losses. One should not expect CFD simulations to perfectly predict these losses, especially since the fan blades are not modeled, but as the fan characteristics are obtained with a fan shroud, the effect of the fan shroud is more or less taken into account twice. Thus, the fan characteristic curve should not include losses over the fan shroud. If this type of data does not exist, one could estimate these losses in simple CFD simulations of a virtual rig. The losses through the fan shroud are then subtracted from the measured fan characteristics. Typical effects of this are shown in Figure 12.

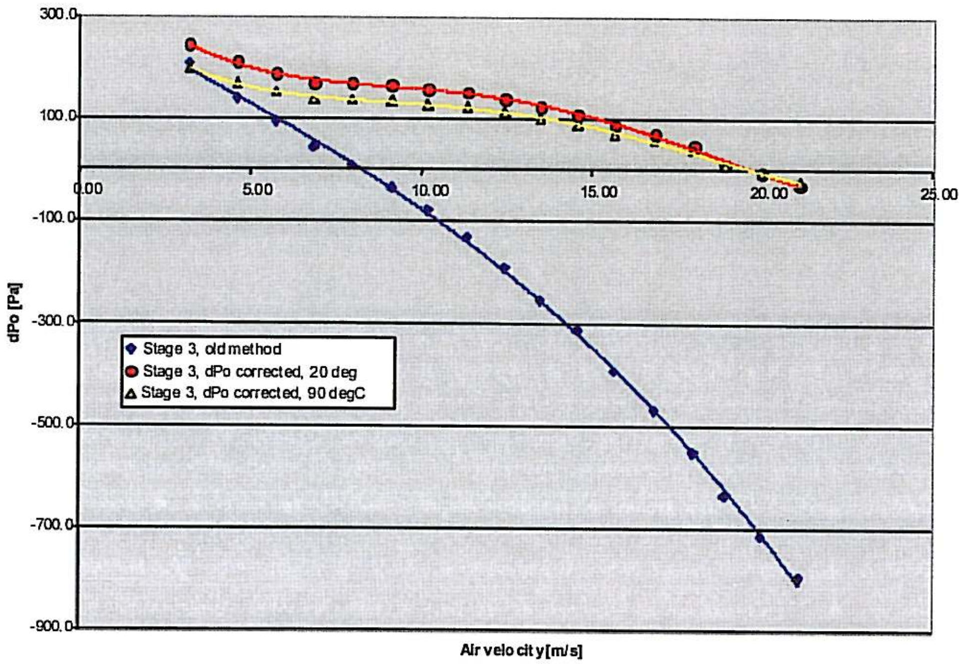

Figure 12: Fan characteristics as received (blue), corrected for fan shroud losses (red), and corrected for fan shroud losses and 90 °C air temperature (yellow)

Also, since the fan characteristics depend on the air density in the fan blades region, the curve should be correlated to the air temperature at the estimated working conditions in the vehicle.

4.2.4 Comparison with measurements

An extensive program was launched to establish how the CFD method compared to measurements. A Volvo S80 T6 and a Volvo V70 N2S were used for comparison.

For both the S80 and the V70 at 15 different run conditions, computed data was compared to experiments. The radiator top water temperature obtained with FLUENT CFD method 3 was within ± 4 °C of the experimental data with a trend that the difference is dependent on the vehicle speed. This, of course, only holds for when high quality boundary conditions are available. The cooling airflow rate was judged to be predicted with approximately 10 % accuracy. The stated accuracy holds for medium to high velocities, while idling and low velocities exhibit severe problems both for measurements and computations.

5. CONCLUSION

FLUENT has been used extensively all through the Volvo XC90 project for cooling performance development. Ranging from fast 2D computations giving generic answers in the beginning of the project to 3D computations using high geometric detailing. The main evaluation parameter throughout the project has been cooling air mass flow, but velocity distribution on heat exchangers and streamlines have been used to give recommendations and answers to the project. The target of 50 % more cooling airflow for the Volvo XC90 compared to the Volvo S80 was successfully accomplished.

Along with the development of the cooling performance for the XC90, the CFD methodologies have been developed. This is partly as a consequence of project demands for answers to target parameters as radiator top water temperature, but also as the CFD group's ambition to offer state-of-the-art methods. FLUENT was used to couple the cooling air flow side to the hot side of the heat exchangers through the use of User Defined Function programming. Comparisons with experiments verified the underlying theory and showed good accuracy of the method. CFD method 3 is now currently used in all projects where cooling performance is addressed.

6. APPENDIX 1: Short summary of solution methodology for CFD methods 1-3

The description of the different CFD methods below attempts to show the unique features for each method. Thus, common features as choice of turbulence model, discretisation scheme, etc have been left out.

6.1 CFD method 1

- Solve the momentum- and turbulence equations until convergence.
- Model the effect of the heat exchanger on cooling airflow using porous media and pressure drop correlation.
- Output parameter is cooling air mass flow.

6.2 CFD method 2

- Solve the momentum and turbulence equations until convergence.
- Empirical pressure drop correlation for individual heat exchangers should preferably come from working conditions. In reality, the transfer of heat from the hot side of the heat exchanger to the cooling air leads to a change in air density and temperature. Its effect on the pressure drop will then to some extent be taken care of.
- Solve only the energy equation using a frozen velocity and turbulence field together with the UDF's.
- Input needed for an individual heat exchanger is heat rejection rate, heat transfer performance curves and hot side media flow rate.
- Output parameters are isothermal cooling air mass flow rate and radiator top water temperature.

6.2 CFD method 3

- Solve the momentum and turbulence equations for approximately 400 iterations. This gives a good flow field distribution although not necessarily complete convergence.
- Continue to solve the momentum, turbulence and energy equation simultaneously using the UDF's for the heat exchangers.

- Air density effects are taken care of through the UDF's together with the simultaneous solution of all equations. Thus, empirical pressure drop correlation for individual heat exchangers should be expressed at adiabatic conditions.
- Fan shroud influence on fan performance curve should be removed.
- Output parameters are cooling air mass flow rate from working conditions and radiator top water temperature.

REFERENCES

[1] Walker, T. "Some Aspects of the Aerodynamic and Thermodynamic Development of the new Volvo XC90". International Stuttgart Symposium, Automotive and Engine Technology, 2003.

[2] Foss, J. et.al. "Evaluating CFD Models of Axial Fans by Comparisons with Phase-Averaged Experimental Data". SAE Technical Paper 2001-01-1701, VTMS Congress, Tennessee, USA, 2001.

[3] Jerhamre, A. "Linking front cooling air flow and heat rejection in cooling package using UDFs". Volvo internal report, 2000.

[4] Jönson, A. "Influence on boundary conditions on the method 'Cooling package user defined function' in Fluent". Volvo internal report, 2001.

[5] Jerhamre, A. "On general knowledge of heat exchangers". Volvo internal report, 2001.

[6] Jerhamre, A. "Cooling package UDF version 3.1: Theory, verification and usage". Volvo internal report, 2001.

[7] Jerhamre, A. "Comparison of THDCAE02 and THDTUNEL: Front cooling air flow and top hose water temperature simulations". Volvo internal report, 2001.