CFD as a Part of a CAE driven Development Process Experiences from the Automotive Industry

Walter Bauer
DaimlerChrysler AG, Stuttgart, Germany

Abstract

This article describes how CFD is embedded today in the vehicle development process at the department of Mercedes-Benz passenger car development. Through examination of several CAE applications including Underhood Flow Analysis/Vehicle Cooling, Passenger Compartment Flow/Air Conditioning/Thermal Comfort, Brake Cooling/Underhood Component Temperature Analysis and External Flow/Aerodynamics we illustrate how different the specific requirements are for optimal use of CFD.

Introduction

Since the first real industrial Computational Fluid Dynamics (CFD) applications in the automotive industry about twelve years ago, the vehicle development process has changed dramatically. The introduction of CAD in the vehicle development process about 15 years ago had a big impact on this process. It was the basis for shorter turn around cycles of CAE applications in general. It made the development of powerful mesh generation tools possible. These preprocessing tools together with the continuously increasing performance of the computing hardware and software led to more and more complex applications. The idea of *virtual vehicle development* was born.

Mercedes-Benz Development System

The Mercedes-Benz Development System (MDS) - a vehicle development process - is as all vehicle development processes very complex and repeats for every new vehicle (see figure 1). For the purposes of this article and to keep it simple we subdivide the vehicle development process into three parts: the Concept Phase which concludes with concept specifications, the Design Phase which concludes with the design specifications and the Initial Phase (minted by vehicle prototyping and testing) which concludes with the Product Launch. We also focus our view on CAE and CAD activities. Currently the vehicle development process is subjected to many changes. The Concept and Design Phases, especially, are more and more driven by CAD and CAE and at the same time the overall process is shortened. The idea behind all these changes is to have at the end a vehicle development process in which all concepts and designs are reviewed by means of CAE and CAD, i. e. all concepts are suitable for production, the vehicle packaging is collision free and producible, and in addition all vehicle functions fulfill the technical specifications.

The realization of a process like MDS requires that all CAE applications fit into their given time frame. Usually there is more than one analysis loop necessary to optimize a vehicle function. Therefore all parts of an analysis loop (geometrical data collection, preprocessing, processing, post processing) (see figure 2) must perform very well and the quality of the results have to be excellent. Because the process is so cyclic, it is worth putting a lot of effort into optimizing the different analysis loops with the criteria of minimal turn-around time, best necessary geometrical resolution and best result. Moreover it has to be guaranteed that comparisons of the CAE results between different geometrical variations within one vehicle development process and other vehicle development processes do not lead to wrong decisions.

Preliminary Remarks

In the following it is shown how CFD is applied today at the department of Mercedes-Benz passenger car development in Stuttgart for the so-called full vehicle functions like underhood flow/vehicle cooling, passenger compartment flow/air conditioning/thermal comfort, brake cooling/underhood component temperature analysis and external flow/vehicle aerodynamics. The meshing is always based on CAD data. For the collection of corresponding up-to-date CAD data a database of these CAD data should exist to avoid CAD data collecting by calling the responsible design departments.

Before CFD was introduced in the MDS as a development tool, comprehensive studies (for all applications) were completed and are still going on to provide security of the CFD results. For more details please refer to the chapter References.

Before CFD was able to became an essential part of the MDS two requirements needed to be fulfilled:

- The results of the CFD analysis have to meet the application specific expectations.
- The results must be available within the given timeframe.

Even excellent results provided too late are more or less worthless.

Resources

The software for solving all these vehicle flow phenomena is STAR-CD and the hardware used are an IBM SP2 (80 CPUs) and a SGI ORIGIN 2000 (32 CPUs). The IBM SP2 is applied to compute the flow problem and the SGI ORIGIN 2000 to do pre- and post processing. All workstations are equipped with a minimum of 2 GB RAM.

Working out all full vehicle functions requires manpower of about 25 man years per year. Additional in-house manpower is needed (2 - 3 years) to improve the applied techniques and to introduced new ones.

Underhood Flow Analysis/Vehicle Cooling

The simulation of the underhood flow by means of CFD started at Mercedes-Benz in 1990. Since that time for more than fifteen different model series (A-, C-, E-, S-Class, etc) three dimensional CFD simulations for underhood flow have been performed. For each model series there is always more than one engine variation analyzed. Today for all vehicles under development three dimensional CFD simulation is together with a one dimensional coolant flow simulation exclusively used to provide security of the cooling concept. A big effort was necessary to reach this high level of application (see [1] and the references listed there).

Optimization loop

Along the MDS there are four major geometrical mesh up-dates and between these up-dates there are several optimization loops. Figure 2 shows the generation procedure of a basic mesh or major up-date. The biggest parts of this procedure are the CAD data collection and preparation and the mesh generation itself. Smaller geometrical modifications need only about one week. The first mesh generation requires collection of about 200 CAD files of the total size of about 1.7 GB. This amount of data can be reduced by throwing away unnecessary details. This results of 600 MB data in VDA format. For an optimization loop the analysis time and the post processing time remain the same. Within an optimization loop typical geometrical modifications are the removal or optimization of the front grill, bumper opening inserts, size, position and arrangement of the heat exchange package.

Mesh requirements

The mesh size increased in the last ten years from 600 000 up to 4 - 5 million elements. The demands of the mesh structure and on the flexibility of geometrical changes are manifold. The mesh topology remains for all vehicles the same (see figures 3 - 5) and it is the result of a long and still ongoing improvement process. The mesh modification for minor geometrical changes are made only locally. The differences in the cooling air mass flow rates can sometimes be small and therefore it is very important that these results are mainly caused by the geometrical differences and not by other effects like global mesh influences.

For the simulation of the cooling fan three fan models are available [2]–[4]: a body force model, a multiple reference frame model and a sliding mesh model. For the cooling simulation in most cases the body force model works fine. If the exact flow structure and temperature distribution downstream of the fan must be known the more complex models are required.

The mesh boundaries have to be far enough off the vehicle surface especially in front of and behind the vehicle to avoid influence on the flow results.

Today all meshes are generated with the help of the preprocessors EZUhood and pro*am. The post processor EnSight is applied for the visualizations of the three dimensional flow phenomena at the power wall (Virtual Reality).

The CFD analysis is at present performed with a minimum of eight processors per job. The turn around time of the job with a mesh size of 4 –5 million elements for a so-called cold analysis (temperature equation is not solved) using eight processors is around 8 - 10 hours. The corresponding time for a so-called warm analysis is between 20 – 24 hours.

Load Cases

For the CFD analysis there are usually three standard load cases applied:

1) Vehicle driving with maximum speed and cooling fan off
2) Vehicle driving uphill with trailer towing and cooling fan on
3) Vehicle with no speed, engine idle, air-condition system full load, cooling fan on

For all load cases the heat exchanger cores are modeled by means of so-called porous media, the wheels are spinning with corresponding rotational speed and the ground is moving also with corresponding speed.

Results

The results of the three dimensional underhood flow analysis is used for the prediction of the cooling air mass flow rates through all heat exchangers and the cooling air velocity distribution across all heat exchangers. The flow structure in the underhood region is also investigated especially in the front-end section. Additionally if temperature boundary conditions are applied the temperature distribution in the underhood region is also available. Again, it is important to say that the cooling air mass flow rates through the heat exchangers alone does not provide security for the cooling concept. The most important parameter is the coolant temperature which can be calculated with the help of a zero or one dimensional simulation tool e.g. Flowmaster.

Passenger Compartment Flow/Air Conditioning/Thermal Comfort

Around 1998 an approach called TEKOS was implemented in the MDS to predict the thermal comfort of passengers. TEKOS consists of a CFD program and two other simulation tools, a program called SWF to simulate the heat transfer inside the passenger compartment and the heat exchange between the exterior and interior and a program called TIM to simulate the thermal properties of the passengers (see figure 6 and [5]). The CFD analysis provides the heat transfer coefficient distribution, temperature distribution and flow structure inside the compartment (see figures 7 - 9).

Since 1998 the thermal comfort of around 10 model series is simulated by TEKOS. Nowadays TEKOS is applied routinely for all model series in the MDS and provides security of the air conditioning concept.

Simultaneously to the introduction of TEKOS the use of CFD for the flow analysis inside the ventilation ducts, the heater/ air condition box, water separation box etc. started.

Optimization loop

There is one major geometrical up-date for TEKOS besides the basic mesh generation within the MDS. The optimization loops are executed to rate a given positioning of ventilation louvers with regards to thermal comfort for a specific set of boundary conditions (solar load, outside temperature, air humidity, etc.). At development time there are in total around 30 – 40 optimization loops (minor geometrical changes only). The first optimization loop starts at Concept Phase providing security for the concept specifications. Most work is done at Design Phase. Parallel to the analysis of the thermal comfort, CFD is also applied to optimize the ventilation ducts with regards to pressure losses and to the heater/air condition box with regards to pressure loss, mass flow rate and temperature distribution across the box outlets.

Mesh requirements

The all hexahedral mesh for the CFD analysis exists of about 1.5 – 3.5 million elements depending on the vehicle size (see figure. 10). The mesh structure must allow the user easily to change the positions of the ventilation louvers. For the first concept studies the ventilation louvers are modeled as inlet boundary conditions. Later, after the major geometrical up-date, the mesh of the ducts plus the mesh of the ventilation louvers are added to the mesh of the passenger compartment.

All meshes for TEKOS are made by means of ICEM-HEXA. The CFD analysis part of the TEKOS procedure is performed with eight processors per job. The CFD part is subdivided into ten times 100 iterations. The turn around time for each 100 iterations for a mesh size of 2.5 million elements is about one hour.

The meshes for the deicing and defogging analysis are based on the TEKOS-CFD-mesh. Element layers for the window glazing are added.

Load Cases

For the CFD analysis the following standard load cases are applied:

1) Winter cases: Steady state and transient (heat up)
2) Summer cases: Steady state and transient (cool down)
3) Windshield deicing
4) Defogging

For load cases 1 and 2 (steady state only) and a given position of ventilation louvers the inlet mass flow rate, the blow out directions and air temperatures are varied until the optimal thermal comfort for all passengers is achieved. For the transient cases only the inlet air temperature varies in time, the other boundary conditions remain the same. The transient air temperature is calculated with the help of one dimensional simulation tools (refrigerant and coolant loop). For the airside of the condenser the calculated air mass flow rate from the underhood flow analysis is applied.

Results

The results (heat transfer coefficient distribution) of the three dimensional passenger compartment flow analysis are used within the TEKOS procedure as input data for the programs SWF and TIM. The TEKOS procedure provides the thermal comfort diagram (see figure 7) for each passenger.

Brake Cooling/ Underhood Component Temperature Analysis

Brake Cooling

In 1992 the first steps were done to simulate the brake cooling. This type of simulation requires first a simulation of the flow inside the wheel arch and second a heat transfer analysis of the brake components. The heat transfer analysis is done by means of the Finite Element Program ABAQUS. A procedure called ALABASTA (for details see [6] – [8]) was developed to transfer data (heat transfer coefficients) between STAR-CD and ABAQUS. So far seven different model series has been investigated. The ALABASTA procedure is nowadays routinely applied to all new vehicle developments.

Optimization loop

The optimization starts with a mesh based on the predecessor geometry to study new concepts (see figure 11). Later during the Design Phase the latest geometry of the front axle and brake assembly is used to create a mesh up-date. In total there are about ten optimization loops based on this up-dated mesh i.e. removing the cover plate, changing the position of the brake caliper.

Mesh requirements

The CFD mesh consists of around 3 million elements. The current mesh topology reflects the needs of the optimization loops and makes the variation of the mesh easier. However getting this mesh takes longer compared to the quickest mesh. The CFD mesh is created by means of pro*am (samm elements). All rotating components are computed using a Multiple Reference Frame (MRF) analysis. The mesh is a refined part of the full underhood mesh.

Load case:

For CFD:
Steady state solution for 140 km/h

For FEM:
1) Alpine downhill braking
2) Breaking Power/Distance test (Cyclic braking and acceleration maneuvers)

Results

Besides the heat transfer coefficient distribution the CFD analysis indicates how the air in the wheel arch has to be guided to best cool temperature sensitive components (see figures 12 and 13). To get the heat transfer coefficients for other vehicle speeds the *power law* is used.

Underhood component temperature analysis

At the moment two methods are tested for the temperature analysis of underhood components. Firstly the same procedure which is applied to the brake cooling (ALABASTA) and secondly a more general approach based on MpCCi. The main desire is greater flexibility in the mesh generation process. In the future we have to deal with the situation, for example, that a supplier of an electronic component would like to use the distribution of the heat transfer coefficients calculated by CFD for the heat transfer analysis of its component. We have not yet finished a complete optimization loop for this application. An example (engine mounting) is shown in figure 14 and 15).

External Flow/Vehicle Aerodynamic

The external flow analysis of a vehicle by means of CFD is not yet fully implemented in the MDS. First steps towards the full implementation are made. For vehicles with a smooth underbody and closed underhood excellent results for drag and lift could be achieved. For more detailed underbody geometry additional effort is necessary to improve the results. For better comparison of measured and computed CFD results three wind tunnels were analyzed by CFD (see [9] and [10]).

Optimization loop

In the Concept Phase the optimization loop is based on simple vehicle surfaces meaning smooth underbody, simple wheel geometry, no side mirrors, no gaps, no underhood. If the surface data are of good quality the timeframe to do one loop is about one week. Surface preparation and mesh generation takes 2 – 3 days and the turn around time for the CFD-analysis is about 2 days.

Mesh requirements

For a vehicle with smooth underbody the mesh requirements are about 4 – 6 million elements for half a car. The mesh structure is very complex and the result of very intensive study and we are continually working to improve it (see figure 16). The mesh is always generated with the help of EZAero and pro*am.

The CFD analysis is usually performed with a minimum of 16 processors. The turn around time for a mesh size of 5 million elements is about 36 hours. For the pre- and post processing the SGI ORIGIN 2000 is used.

Results

As already mentioned the results (lift and drag) for vehicles with a smooth underbody are excellent. Even small changes in the vehicle shape are predicted very well. The visualization of the three dimensional flow (Virtual Reality) helps in understanding the flow structure (see figures 17 – 19).

Conclusions

It was shown that in most cases (except the external flow/vehicle aerodynamic) CFD is embedded in a procedure with other simulation tools. The CFD results alone do not fulfill the MDS requirements for the full vehicle functions, only the combination with other simulation tools provides the full range of information. Therefore all parts in the simulation chain must deliver accurate results. To be on the safe side a lot of testing and comparing must be done to find the best combination of tools for a specific optimization loop.

Experiences from the past show that the complexity of the simulation models and the number of optimization loops increase all the time. For some applications like thermal comfort there are process requirements for interactive CFD application to be able to discuss on-line the results of variations. In the future more transient CFD simulations will be necessary. Therefore to increase the today’s high level of CFD application and to meet future MDS requirements the following need to be done:

- The optimization loops can only be shortened significantly if the meshing phase is shortened dramatically.

- The turn around time of the CFD analysis must be shortened therefore the robustness and convergence rate of the code must be improved.

- More complex CFD applications require suitable turbulence models.

References:

[1] T. Binner, M. Cigarini *A Study of Experimental and Computational Tools for Automotive Cooling Design*, 4th World Fluid Dynamics Days ’98, Conference Proceedings

[2] Spindler, T.,Stetter, H., Reister, H.,*'Numerical Simulation and Measurement of a Vehicle Cooling Fan at a Fan Test Rig'*, 4th VTMS Conference, C543-70, IMechE and SAE, London, UK, 1999

[3] Spindler, T, .Stetter, H., *'Numerische Simulation und Messung der Einflüsse des Kraft-*

fahrzeuglüfters auf die Motorraumdurchströmung‘ VDI Band 1559: Berechnung und Simulation im Fahrzeugbau, VDI Verlag, 2000

[4] Spindler, T. *'Numerical simulation and measurement of the vehicle cooling fan’s influence on the engine compartment flow‘,* StarCD Underhood Thermal Management Seminar, Nürnberg, Germany, 2000

[5] Juergen Maue, Dietrich Wahl, Joachim Currle: "*Computation of the Thermal Environmen in Passenger Compartments and Evaluation of Thermal Comfort";* 4th International Conference and Exhibition "Comfort in the automotive industry recent developments and achievements"; Bologna, 6-7 October 1997

[6] Krüsemann, R. / Schmidt, G. *Analysis and Optimization of Disk Brake Cooling Via Computational Fluid Dynamics*, SAE Technical Paper Series No. 950791 (1995)

[7] Krüsemann, R. / Urban, J. / Zajcev, V. *Brake Disk Temperature Calculation in ABAQUS Based on Heat Transfer Coefficients from Numerical Flow Simulation*, ABAQUS Users’ Conference Proceedings, Milano, Italy (1997)

[8] Hopf, A. / Gauch, A. *Numerische Simulation der Bremsenkühlung mit CFD und FEM*, VDI-Berichte No. 1559, Düsseldorf, Germany (2000)

[9] Jehle, E. *Numerische Simulation und topologische Untersuchung eines 3D-Strömungsfeldes um ein realistisches Fahrzeug unter Berücksichtigung der Kühlluft*. VDI, Berechnung und Simulation im Fahrzeugbau, (1559), 2000

[10] Jehle, E. *Strömungsstrukturerkennung auf der Basis von numerischen 3D Navier-Stokes-Simulationen zur Einbindung in den Fahrzeugentwicklungsprozess*. PhD thesis, Universität Karlsruhe (TH), will be published in 2001

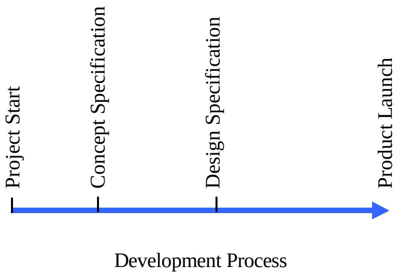


Figure 1: Simplified Development Process

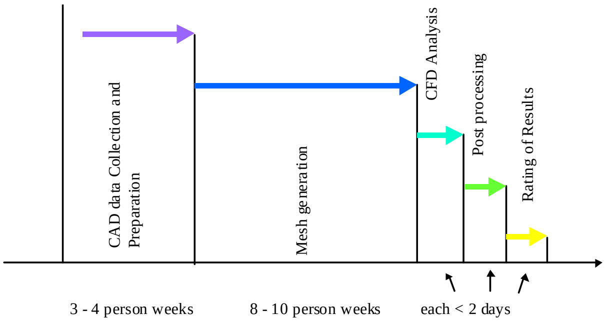


Figure 2 : Initial Optimization Loop *Underhood Flow/Vehicle cooling*

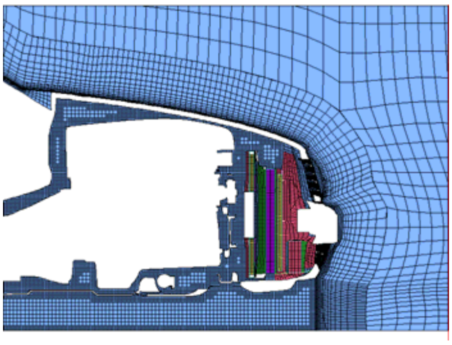


Figure 2: Mesh Structure for Underhood Flow/Vehicle Cooling

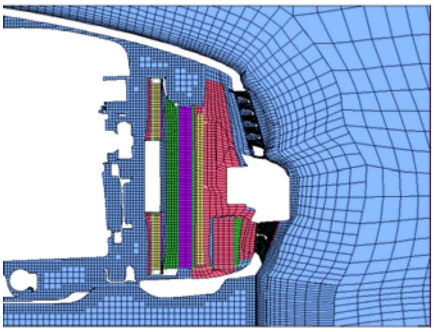


Figure 3: Mesh Structure for Underhood Flow/Vehicle Cooling (Detail)

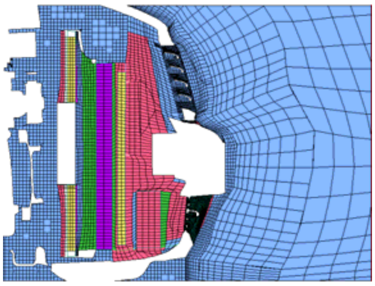


Figure 4: Mesh Structure for Underhood Flow/Vehicle Cooling (Detail)

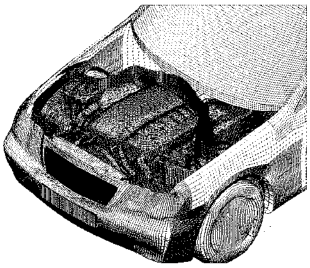
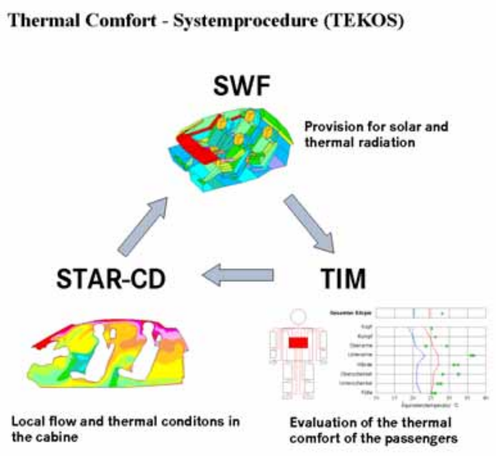
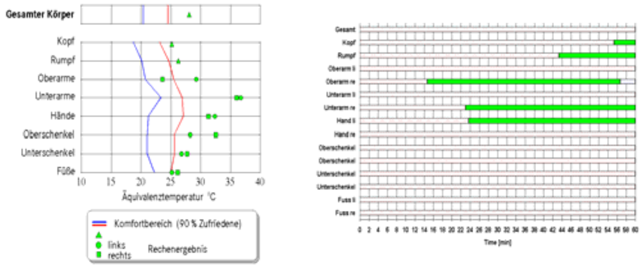


Figure 5: Mesh Structure for Underhood Flow/Vehicle Cooling





Steady State

Transient

Figure 6: TEKOS Procedure

Figure 7: TEKOS Thermal Comfort Diagrams

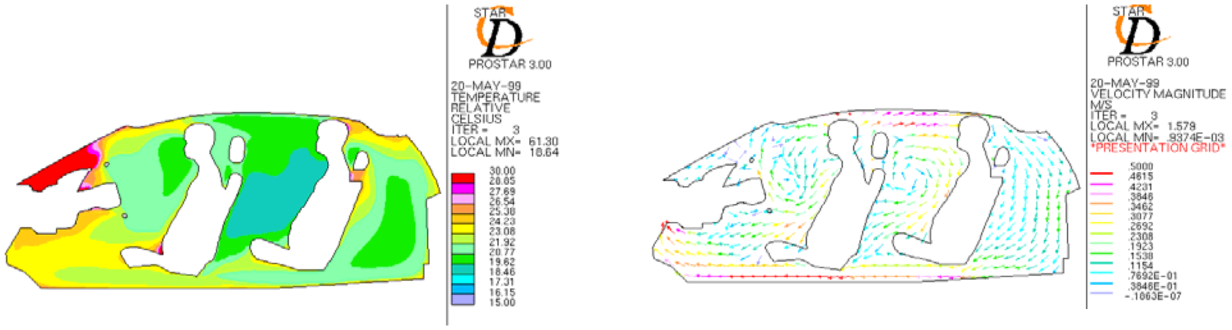
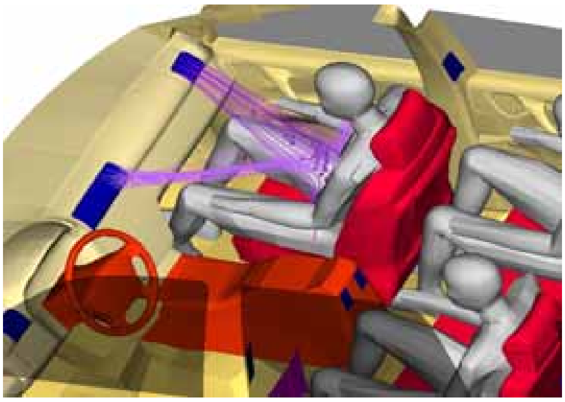


Figure 8: TEKOS Results, Temperature Dustribition and Flow Structure



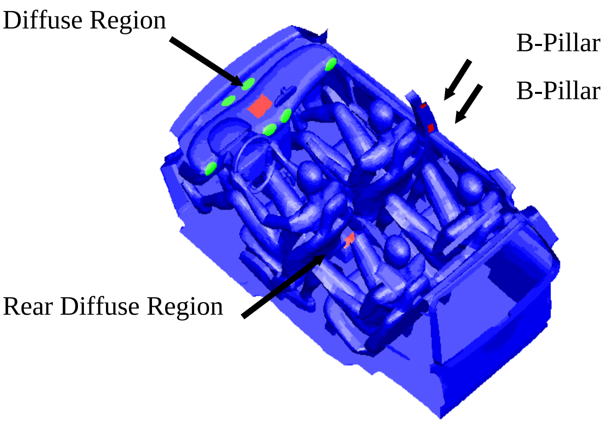


Figure 9: TEKOS Results, EnSight Animation

Figure 10: TEKOS

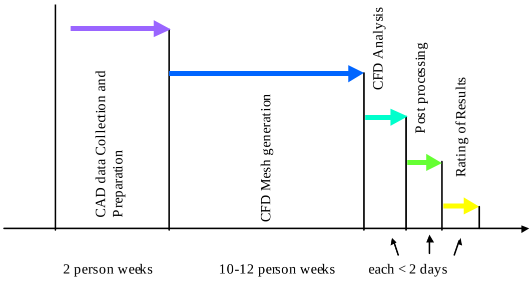


Figure 11 : Initial Optimization Loop
Brake Cooling

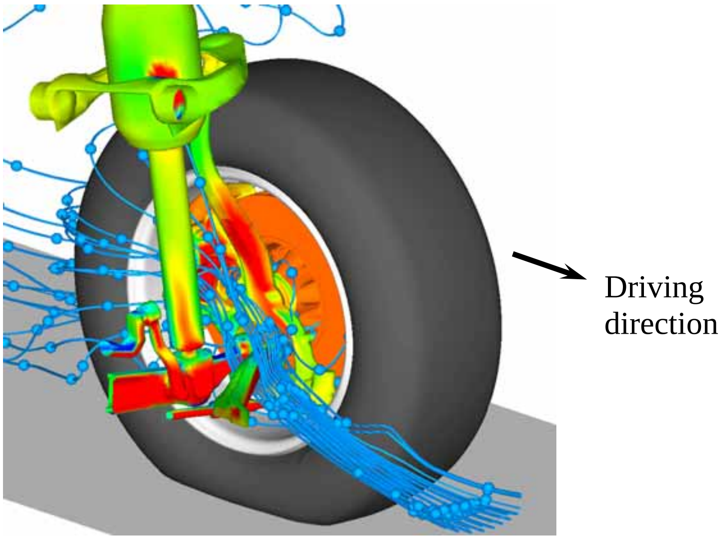


Figure 12: CFD-Results of Brake Cooling
Analysis

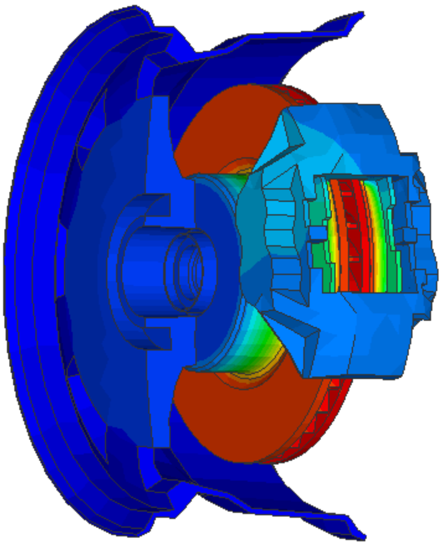


Figure 13: FEM-Results of Brake Cooling
Analysis

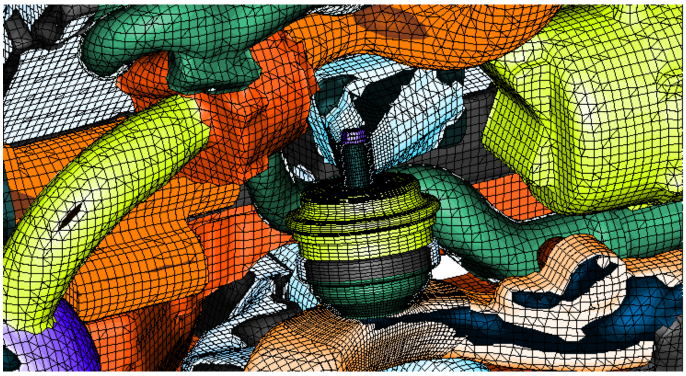


Figure 14: Surface Mesh around an Engine
Mounting

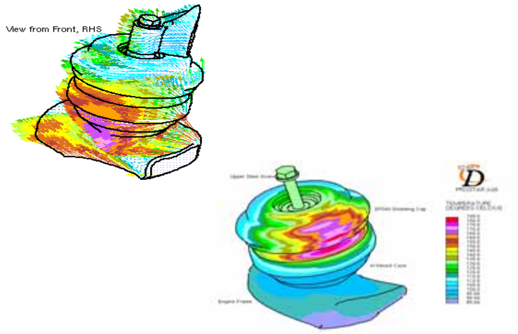


Figure 15: Results of Flow-Heat Transfer Interaction Analysis

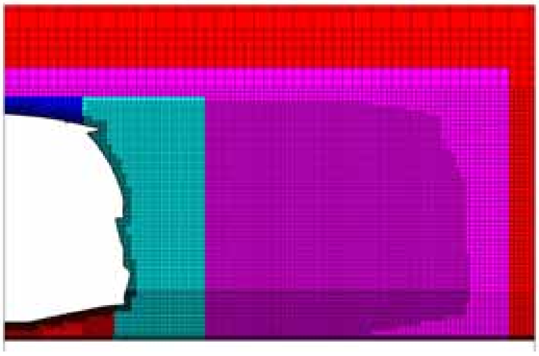


Figure 16: Mesh Structure of an Aerodynamic Mesh

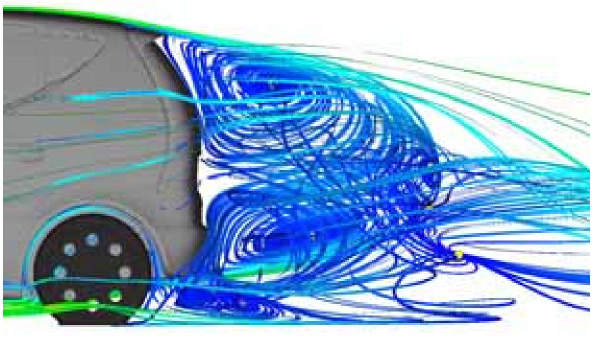


Figure 17: Wake Flow Structure

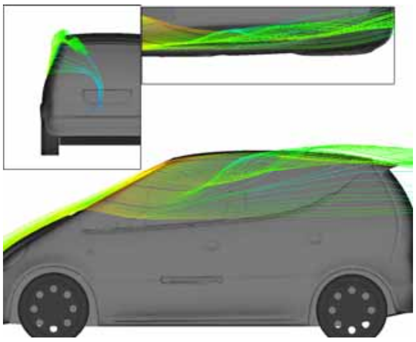


Figure 18: A-Pillar Vortex

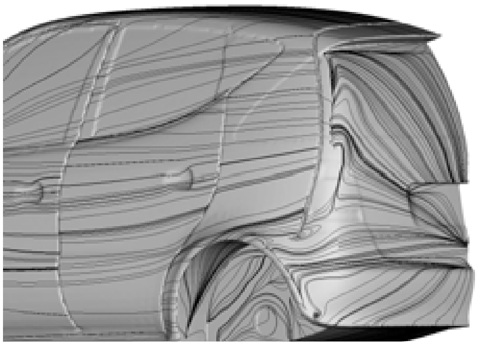


Figure 19: Near Wall Streamlines