EU-BRITE-EURAM Project

Models for Vehicle Aerodynamics (MOVA)

Project No. BE97-4043
Contract No. BRPR-CT98-0624

SYNTHESIS REPORT
FOR PUBLICATION
July 2002

Coordinator/Editor: K. Hanjalic

Project funded by
THE COMMISSION OF THE EUROPEAN COMMUNITY
under the Industrial & Materials Technologies Programme
(BRITE-EURAM III)

SYNTHESIS REPORT
FOR PUBLICATION

Project No: **BE97-4043**

Contract No: **BRPR-CT98-0624**

Title: **Models for Vehicle Aerodynamics (MOVA)**

Coordinator: TU Delft (Prof. K.Hanjalić)
Partners: UMIST (Prof. B.E. Launder)
LSTM (Prof. F. Durst/H.Lienhart)
EDF-LNH (Prof. D. Laurence)
AVL (Dr. B. Basara)
PSA (Dr. L. Elena/J-L. Aider)

Reference period: from 1.4.1998 to 30.9.2001

Starting date: 1.4.1998 Duration: 42 months

Date of issue of this report: July, 2002

Project funded by The Commission of the European Community
under the Industrial & Materials Technologies Programme
(Brite-EuRam III)

Contents

1. Summary

Keywords: *Vehicle aerodynamics, Turbulence modelling, Computational Fluid Dynamics*

The prospects of designing a car or other vehicle shape using solely computer simulation without performing wind tunnel experiments on a scale model, has long been a challenge for car manufacturers. Nowadays many, if not all, car manufacturers use Computational Fluid Dynamics (CFD) packages. However, although viewed as potentially a major tool for design and optimization in many branches of industry, CFD has not yet reached the level of reliability and confidence where it could be used exclusively as a design tool. One of the major difficulties is the proper simulation of turbulence. Because of the great complexity of the car geometry and the still limited computational resources, numerical simulations by the car industry are based on solving the averaged equations of fluid motion (Reynolds-averaged Navier-Stokes, "RANS", approach), which involve different approximations ("modelling") of various flow and turbulence phenomena. In the automotive industry, expensive and time consuming large-scale wind tunnel measurements are still unavoidable for the final shaping of the automobile body, whereas the CFD assists in the preliminary development phase and for qualitative exploration of effects of various design parameters. A reliable CFD tool, if available, would reduce both the costs and time of the research and design, and accelerate the appearance of new designs on the market. Hence, the further improvement of CFD is of primary interest to all automobile industries. The goal of the MOVA project (Models for Vehicle Aerodynamics) was to develop a computational turbulence model (or models) for the road-vehicle industries, which should make it possible to perform accurate and efficient computations of external flow over road vehicles. With such a model, the CFD codes are expected to become a reliable tool giving relatively inexpensive and fast computations in the design process as well as to opening a route for interactive CFD - wind-tunnel optimization.

The objectives were pursued by the joint efforts of six partners, three from academia and three from industry. The research tasks consisted of developing, refining and validating advanced RANS models, supplemented with new experiments and Large Eddy Simulations (LES). The optimal turbulence model was sought among three model classes: nonlinear eddy viscosity models (NEVM), three-equation elliptic-relaxation eddy-viscosity models (ERM) and the differential second-moment (Reynolds-stress) closure models (DSM). Some new simple improvements of the standard linear eddy-viscosity model (EVM), as well as a new hybrid EVM/DSM (HTM) have also been derived and tested. Besides model refinement, new wall functions were also developed, which should permit the use of high-Reynolds-number model versions in complex flows.

The focus of the development was especially the rear end of a car body, which greatly affects the vehicle drag. Model refinements were validated in several generic test flows relevant to vehicle aerodynamics where experimental data exist. New experimental data were also generated in two dedicated experiments: detailed mean flow and turbulence measurements in flow over a generic car afterbody (Ahmed car model) in a laboratory wind tunnel, and velocity and pressure measurement in flow over a simplified 1/5 scale car model (PSA model) in an industrial wind tunnel. The final validation of the developed models in the computation of flows over both simplified car models (Ahmed and PSA models) showed notably improved agreement with experiments, as compared with the standard linear eddy viscosity models.

In parallel to the RANS model development, a new unstructured finite-volume solver for Large Eddy Simulations (LES) was developed and used to simulate flow over a simplified car mirror. The LES code is envisaged as an additional tool for optimizing the shapes of car mirrors and other vehicle protrusions, as well as to generate input for computing aerodynamic noise.

2. The Consortium

(TUD) - Delft University of Technology, Department of Applied
Physics, Section Thermal and Fluids Sciences,
Lorentzweg 1, 2628 CJ Delft, The Netherlands .
Contact person: Prof. K. Hanjalić (project coordinator),
Tel: +31 15 278-1735, Fax: +31 15 278 1204, E-mail: hanjalic@ws.tn.tudelft.n

(UMIST) - University of Manchester Institute of Science and
Technology, Department of Mechanical, Aerospace and Manufacturing
Engineering, P.O. Box 88, Manchester M60 1QD, United Kingdom
Contact person: Prof. B.E. Launder,
Tel: +44 161 200 3701, Fax: +44 161 200 3723, E-mail brian.launder@umist.ac.uk

(LSTM) - Friedrich-Alexander University, Erlangen-Nuernberg,
Lehrstuhl fuer Stroemungsmechanik,
Cauerstrasse 4, 91058 Erlangen, Germany
Contact person: H. Lienhart,
Tel: +49 9131 852 9501, Fax: +49 9131 852 9503, E-mail: lienhart@lstm.uni-erlangen.de

(EDF) - Electricité de France, Lab. National d'Hydraulique,
6 Quai Watier, 78401 Chatou Cedex, France
Contact person: Prof. D. Laurence
Tel: +33 1 3087 7257, Fax: +33 1 3087 8086, E-mail: dominique.laurence@edf.fr

(AVL) - AVL-List GmbH, Computational Fluid Dynamics,
Hans List Platz 1, A 8020 Graz, Austria
Contact person: Dr. B. Basara
Tel: +43 316 787 705, Fax: +43 316 787 777, E-mail: branislav.basara@avl.com

(PSA) - DRIA/SARA/PVMO/AERO, PSA Peugeot Citroeën,
Route de Gisy, 78140 Velizy - Villacoublay, France
Contact person: J-L. Aider
Tel: +33 1 41 362 877, Fax: +33 1 41 363 177, E-mail: jlaider@yahoo.com

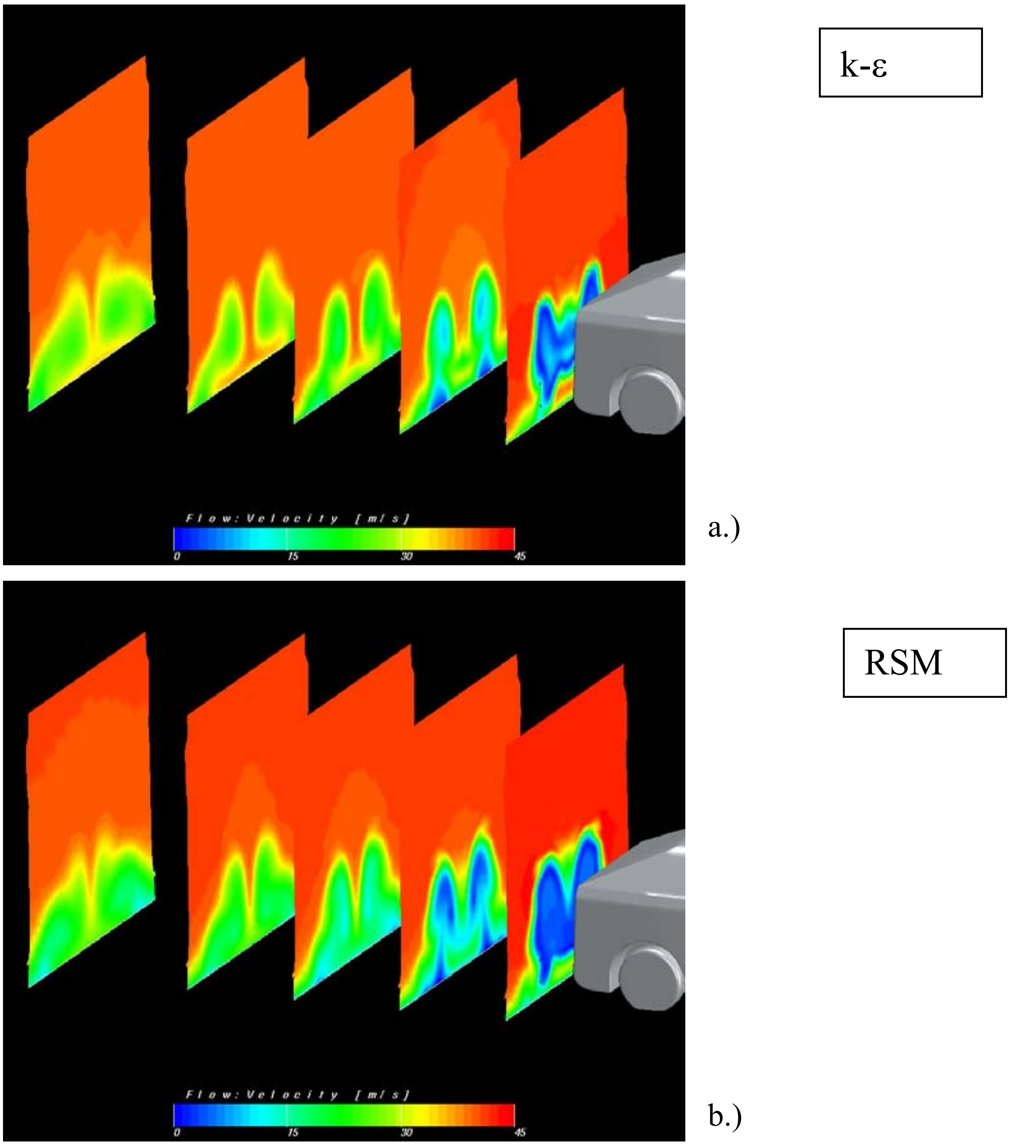

Figure 1. Calculated velocity field in the wake of PSA car model.
(a) k-e model (b) RSM model (AVL calculations).

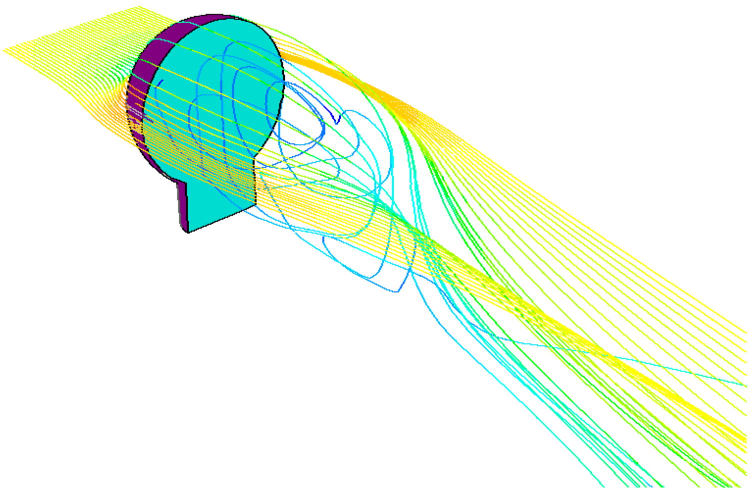

Figure 2. Path lines around a car mirror: Large Eddy Simulations (TU Delft)

3. Technical Achievements

3.1 Industrial Needs

The design criteria for future generations of road vehicles are becoming more and more demanding. Among many targets, two are especially important: meeting the more and more stringent environmental constraints and improving general vehicle performance and safety. Attention is focused particularly on the demands for reduced fuel consumption, reduced noise generated during vehicle movement, and better road behaviour, all of which are strongly affected by the vehicle aerodynamics. Reliable and robust computational tools that can make possible to explore quickly a large number of design options are, therefore, becoming an essential prerequisite for the design and optimisation of the next generation of cars, buses, and trucks. In addition, computational methods are needed to make design faster and more cost-effective in order to better meet the demands from an ever changing marketplace.

Industrial Computational Fluid Dynamics (CFD) codes are being used more and more for these purposes, complementing and replacing in part expensive and time consuming wind tunnel testing. The industry and commercial CFD vendors have jointly defined the main prerequisites for potential extensive industrial application of CFD: reducing the efforts to generate numerical mesh, diminishing the total computation time to acceptable level, and improvement in accuracy. These are also the main targets in the current developments of CFD. Of course, the balance between the above requirements depends on the application and goals. In the design process, very high accuracy is not needed since the task is often reduced to get an insight into the major effects of various design options and alteration. More important is the efficient and fast meshing, low computing time and easy manipulation of input and output data. In research, the role of CFD is different: here the accuracy has the first priority. Of course, the best would be to develop a CFD code that will meet both requirements.

3.2 Challenges to CFD for Vehicle External Aerodynamics

One of the major difficulties encountered in the numerical simulation of industrial flows is the proper simulation of turbulence. While the numerical aspects have been more or less resolved, the turbulence models pose still major uncertainties. Much of the current effort in increasing the confidence in the CFD codes focusses on improvement and further validation of turbulence models. It is generally believed that a class of turbulence models capable of capturing the flow physics relevant to vehicle aerodynamics can be developed if sufficient effort is invested. This would enable a much wider use of CFD in the optimization of the road-vehicle design, that will bring marked rationalization of both time and expenditure.

The automotive aerodynamics is characterized by a number of flow phenomena which are difficult to predict with the conventional CFD codes. Most notable are the flow impingement, unsteady boundary layer separation on curved vehicle body, flow around rotating disks (wheels) and bluff body wake. Flow impingement occurs when the air decelerates in front of an automobile creating a stagnation region. Accurate prediction of the impingement phenomenon is the prerequisite for solving the flow downstream and in the vehicle wake. Energy losses in the wake of a typical ground vehicle were found to make the major contribution to the aerodynamic drag. The location at which the flow separates determines the size, and consequently the drag force. Clearly, a more exact simulation of wake flow and the separation process are essential for the correctness of the overall simulation process. The interaction between a rotating disk and the flow resembles, in a simplified manner, the situation around the wheels and the effects of their rotation on the flow structure. All these (and other) *generic* flows have been selected to serve *as test cases* for validating new models or model modifications. Even if all the above mentioned phenomena are captured with a turbulence model when considered isolated in simple generic flows, this is still not a guarantee that the model will perform satisfactorily when applied to compute the flow around a full car body, even if simplified shapes are considered. Hence, the major test for the new turbulence model should in the end be the accurate predictions of flow over a *realistic car body*, validated against reliable experimental data.

3.3 Project Objectives

The goal of the MOVA project was to develop a workable framework for the road-vehicle industries to efficiently and accurately compute the external aerodynamic flow using validated, improved turbulence models. This should upgrade the performance of relatively inexpensive and fast computations in the design process, as compared to expensive and elaborate wind-tunnel testing, as well as open a route to interactive CFD-wind tunnel optimization. The developed models were to be sufficiently robust, with verified performance, thus becoming a useful tool for the computation of the real-life external aerodynamics of road vehicles.

Three partners from industry and three university research institutions set out to achieve the main project objectives by focusing research effort on the following tasks:

3.3.1 Development and improvement of advanced turbulence models

Three main approaches to the *statistical turbulence modelling within the Reynolds-average-Navier-Stokes (RANS) framework* have been pursued. The first is based on the nonlinear eddy viscosity models (NEVM). This class of models can be regarded as an extension of the widely used and extensively tested two-equation models, which is at present the mainstay of all commercial CFD codes. The second approach utilizes the concept of elliptic relaxation, which has proven to be an efficient technique for accounting for the non-viscous wall blocking effects. Whereas this method can be applied within the framework of the differential (full) stress models (DSM), the final formulation can be significantly simplified through the combination with the two-equation eddy viscosity models - denoted here as three-equation elliptic relaxation models (ERM)). The third route is the refinement of the differential second-moment closure models (DSM) (known also as differential Reynolds stress model), which is regarded as physically the most rational modelling approach. While the computational requirements of the last technique may be significantly higher - both in the computational resources and in the expertise required from the users, the anticipated improvements are expected to justify the costs.

3.3.2 New wall functions

Owing to the fundamental importance of modelling of the fluid flow in the vicinity of solid walls, another objective was to develop *novel wall functions* that will make it possible to achieve accurate flow predictions while still using robust, low-cost high-Reynolds-number versions of the turbulence models.

3.3.3 Generating new data base in dedicated experiments

The prerequisite for calibration, validation and verification of turbulence models for a specific application is the availability of relevant experimental data. For configurations that are relevant to vehicle aerodynamics, such data are scarce in the open literature. For that reason, one of the important objectives of the MOVA project *was to conduct experiments in a research and in an industrial laboratory*. Both studies were aimed at detailed measuring of various flow and turbulence properties, providing thus sufficient information for testing the models and modeling ideas developed in the course of the MOVA project.

3.3.4 Development of unstructured CFD solver for Large Eddy Simulation

In addition to the statistical RANS modelling, Large Eddy Simulation (LES) has been recently gaining in popularity and has been regarded by many as the future industrial tool for accurate prediction of vehicle aerodynamics. LES certainly possess many desirable features and in principle it is superior to RANS. It requires less empiricism and provides information about (large-scale) turbulence spectrum. However, this is still a very expensive technique and in the near future not feasible for computing flow over the whole car body at realistic Reynolds numbers. Nevertheless, LES is already quite feasible for simulating local flows over various car protrusions. For that reason, and in anticipation that the advancement in the computer design and further developments in the LES technique will make this approach more and more attractive for the vehicle industries in the future, one of the objectives of the MOVA project was *the development and testing of a new unstructured computational code for LES* specifically targeting external aerodynamics of car-mirrors and other complex-shaped vehicle protrusions. In addition to making it possible to optimize the design of such vehicle elements, an important application of LES is to provide the input (unsteady velocity and pressure field) for computing *aerodynamic noise* generated by the protrusions.

3.3.5 Validation and implementation of new models

The final objective was to perform computations of flow over a realistic car and to demonstrate the performance of the newly developed models, especially in comparison with the models used in the commercial CFD codes. Extensive validation of the new modelling concepts were to be performed in flows over realistic - though still simplified - car models. For that reason, as well as for the future use of the new models in industrial CFD, one of the important goals was the implementation of the developed models into commercial CF codes used by automotive industry.

3.3.6 Test cases

The considered test cases, used for RANS models validation are:

- Square-sectioned cylinder at different distances from the wall (the mandatory base test case) (TUD/UMIST/EDF/AVL)
- Normally-impinging round jet (TUD/UMIST)
- Plane diffuser (UMIST)
- Rotating disc (UMIST)
- Flow over a backward-facing step (TUD)
- Flow over a periodic hill (TUD)
- Ahmed car model (the mandatory main test case) (LSTM -experiments, UMIST, TUD, EDF, AVL)
- PSA realistic car model (AVL, PSA)

The test flows for LES validation:

- Over a wall-mounted cube (TUD)
- Over an idealized car mirror (TUD)

3.3.7 Summary of deliverables

- Algorithms for the advanced RANS turbulence models: nonlinear eddy-viscosity (NEVM), three equations elliptic relaxation (ERM), and differential second-moment models (DSM);
- Algorithms for the new wall functions to be used in conjunction with any of the above listed (or other) turbulence models
- CFD code for Large eddy simulation (LES) suitable for predicting flows over vehicle car mirrors and other protrusions
- Experimental database for calibration, validation and verification of RANS models in vehicle-relevant types of flow
- Results of testing of the developed models, wall functions and LES in generic flow test cases including simplified car model
- Know-how for the implementation into industrial CFD codes of the developed advanced turbulence models and wall functions

3.4 Industrial Targets

- Improvement of turbulence modellig for the CFD of road-vehicle flows, by demonstrating that the new models predict the basic physics as well as 'real-life' road-vehicle aerodynamics. The aim is to predict the overall aerodynamics quantities, such as the drag coefficient, with a deviation below 5%.
- Implementation of the new models in commercial CFD codes frequently used by the road-vehicle industries.
- To transfer the knowledge about and beyond the new generation of models to the road-vehicle industries and CFD vendors.

With these goals achieved, CFD is expected to become recognized as a reliable design tool within the automobile industry. As a result it should:

- replace up to 50% of the experiments within the 5 years;
- shorten the design-time of a new car to less then 3 years, as currently aimed at by the car industry
- enable the user to quickly address the effect of design parameters on fuel consumption, road stability and safety, and noise generation.
- In this way a reduction of the aerodynamics drag by 20% seems to be feasible within 5 to 8 years.

3.5 Estimated time to market

The developed and validated turbulence models were to be available through their implementation in the computational codes of the industrial partners by the end of the project, and were also to be available to other European research institutes and industries at that time.

3.6 Achievements

3.6.1 Tasks accomplished

The task of *model development and refinement* focused on the analysis, evaluation, and possible refinement of the three classes of models. The NEVM model, developed earlier at UMIST has been further scrutinized in three-dimensional separating flows. A new formulation of the elliptic relaxation for ERM has been proposed, based on direct simulation (DNS) results, which was expected to account better for turbulence scale inhomogeneity and asymmetry near solid walls. Because for complex industrial flows at high Reynolds number the integration up to the wall, as implied by the ERM model, is not convenient, a new approach was investigated in which the ERM model was combined with wall functions. For this purpose a wall function was derived for the elliptic relaxation function. A refinement of the DSM was explored at the quasi-linear level (for pressure-strain) aimed at better capturing of wall non-viscous (blockage) effect. Two lines of approach, both based on direct numerical simulation (DNS) results, were followed, aimed either at redefining the coefficients in terms of turbulence invariant properties, or by implementing a scalar elliptic relaxation equation. A combination of both approaches has also been considered. For complex vehicle flows, both DSM variants have been used with wall functions.

New wall functions have been formulated based on the integration of the turbulence energy production and dissipation over the near-wall control cells. Two novel approaches have been followed. The first approach, called "the analytical wall functions" follows the well known cell-integration method but dispensed with the assumptions of equilibrium wall conditions and the conventional logarithmic velocity law. The second approach, "the subgrid wall functions" decouples the numerical solution of the near-wall cell from the rest of the flow region. A separate solution of the one-dimensional boundary layer equation is performed for each near-wall cell using an embedded one-dimensional mesh.

New dedicated experiments were performed for the selected hatch-back simplified car body (Ahmed model) in a laboratory, aimed at providing detailed experimental data on mean velocity, second moments and third turbulence statistical moments, pressure distribution and surface flow visualization. These data served as input for the computational work, but also as a basis for model validation. Another set of experiments was performed in an industrial wind-tunnel, using a realistic - though still simplified - car model (PSA model), aimed at providing additional data on the pressure and velocity distribution over a realistic vehicle.

All developments have been *validated* in a number of generic flows that included: flow over a square sectioned rod at various distance from a solid wall, flow over a wall-mounted cube, impinging jets, separating flow over a backward-facing step, in a plane diffuser and over a periodic hill (the latter two representing the rear slanted window of a car), flow in a square-sectioned U bend and over a semi-enclosed rotating disc (the latter imitating the flow over wheels) and others. Each of the three modelling partners considered at least three test cases, with one case overlapping between the two partners, and two flows serving as the joint reference cases. The mandatory reference test cases were the above mentioned flow over a square rod, and the flow over the hatch-back simplified car body (Ahmed model) for which one of the partner (LSTM) provided detailed experimental data.

In parallel with the development of RANS models*, new LES code*, based on finite-volume approach with an unstructured grid and fully parallelized, has been developed and tested in several flows, including a surface mounted cube in a matrix, yielding excellent agreement with experiments. The LES code was then used to simulate flow over an idealized *car mirror,* providing aluable detailed information about flow and turbulence in such flows.

The real-life tests and validations of the models were performed by two industrial partners on a specific simplified 1/5 scale vehicle model (PSA model) for which PSA provided experimental data.

3.6.2 Summary of Achievements

The project fills - at least in part - the gap between recent developments in advanced turbulence modelling and the possible use of such models in industrial CFD codes for road-vehicle design. The up-to-date advanced turbulence models have been scrutinized in a number of test flows, the necessary modifications were made in the existing models, new elements have been added and tested, and some completely new modelling approaches have been developed and validated. A better insight into some important flow and turbulence phenomena relevant to car aerodynamics have been gained. A new more accurate treatment of the wall boundary conditions in conjunction with robust models has also been developed. Essential experiments not available in the literature have been provided and used for the validation of the turbulence models. Comparative validations of the models have been performed in flows over two simplified car models, a simpler one (Ahmed model) with all models developed, and a more complex case (PSA model) where only some of the models were tested.

3.6.3 Conclusions on Turbulence Models

The investigation presented above clearly confirm the already known fact that the standard linear k-ε eddy-viscosity (EVM) model with standard wall functions, although being the most widely used model in CFD, has many deficiencies, which come into prominence especially in complex flows. Major deficiency of the standard model is in its inability to reproduce properly flow features encountered in complex flows, such as streamline curvature, impingement, separation, reattachment, secondary motion, rotation, three-dimensionality.

Differential second-moment closure models (DSM) are more superior, though still not perfect. They are also more demanding on computational resources and require better skill to conduct the computations. Two types of DSM's tested in this project (SSG and the elliptic blending model, EBM) both showed visible improvement as compared with the standard EVM. When used with wall functions, the requirements for computing time are about 50-100% higher than for the standard model, which can be considered as manageable. DSM-SSG should be tested in conjunction with the new wall functions, what was not possible in the present project due to time limitation. DSM-EBM, developed in the course of MOVA project, offers some new prospects for capturing better the near-wall stress anisotropy and, moreover, allows the integration up to the wall. While the latter itself is not much popular in industrial CFD because of a significant demand on computation power and time, it is a useful option for flows at lower Re-numbers, but also for a combined use of wall functions and integration up to the wall. The version tested here showed promising prospects, though the development has not been fully completed.

While the DSM type of models will be used more in the future in industrial computations, some intermediate options can bring improvements whilst not requiring excessive computational effort. Two approaches tested, the NEVM and ERM seem especially attractive. The latter being designed for integration up to the wall is not suited for complex high-Re-number flows in its original form, but can be used in conjunction with wall functions. The advantage is that the second scalar variable υ^2 accounting for the non-viscous wall blocking effect even outside the viscous layer, provides better eddy viscosity and accounts at least partially for the stress anisotropy. Better wall functions for this model can still be derived, and we foresee more research in this direction.

Of a number of simple improvements of the standard k-ε model, two new approaches developed in the course of this project, i.e. the "linear-production" (LP) k-ε and the hybrid EVM/DSM (HTM), show both significant potential. The LP model can be especially attractive for industry in the short term because it is very easy to implement into the existing CFD code. The HTM provides a middle route between the EVM and DSM and captures some essential features of complex flows that are intractable by the standard EVM. A shown in the Technical Report, this model is also relatively easy to apply and is sufficiently robust to allow complex computations. It has been successfully implemented in an industrial CFD (SWIFT AVL) code and used without difficulties to compute flow over a real road vehicle including wheels, with only a moderate increase in computing time. It should be noted that HTM can be combined with different DSMs, such as SSG, EBM etc, as well as with different wall functions.

Wall treatment in conjunction with high-Reynolds number closure models has long been the major issue and one of the weakest points in deriving industrial turbulence models.

Industry still cannot afford to perform integration up to the wall when computing external flows over moving real vehicles. Hence, the Wall Functions have long been and will probably remain for some time the only viable option for wall treatment. In this project we provide derivation of several new wall functions. They differ in complexity, but both provided visible improvements. Although further testing and improvement is desired, the functions developed here can already be implemented into the commercial codes. No doubt, each of these will be beneficial and will enhance the accuracy and confidence in the computations.

Finally, what can we recommend? Unfortunately, no definite recommendation can be made, since each of the models tested offers certain advantages and disadvantages. We have chosen to consider models that should meet industrial criteria, i.e. to be easily incorporable into the existing industrial CFD codes and to make it possible to improve their accuracy whilst remaining robust and economical. We have demonstrated that most models and modifications here considered and tested perform better than the standard approach. The final choice remains on the user and will depend on the type of problem, required accuracy and computational speed, and whether the computations are performed for design optimization or for research and development of new vehicle shapes.

3.6.4 Innovative character of the research

The project as a whole has innovative character since it provides the modelling basis for an improved design tool for the road-vehicle industry. The following specific innovations are claimed:

- derivation and or improvement of advanced turbulence models and wall functions for application to external aerodynamics around complex-shaped bodies,
- new accurate experimental data for flow phenomena encountered in car aerodynamics, using up-to-date laser-Doppler and other techniques
- validation of turbulence models in a series of reference generic and real-life test cases
- incorporation of advanced turbulence models in industrial CFD packages, which have been applied to the computation of flow around real car shapes,
- a new unstructured finite-volume computational code for large eddy simulation (LES) of complex external flows relevant to vehicle aerodynamics (also useful as an input for computing noise generation)
- generation of new computational results for real vehicles.

3.6.5 Critical appraisal of the achievement and future prospects

The project as a whole has been successful. All tasks have been accomplished. The automotive industry has gained valuable information about the specific needs for upgrading and improving CFD codes used for research and design. Comparative validation of the new models in two mandatory test cases, as well as in a real-life flow, have provided new information on some long-standing issues that have in the past deterred the use of CFD as a design tool.

Admittedly, not all modelling developments could be implemented in the CFD codes used by the two industrial partners. One of the reason was the delay in providing the final form of the new models developed by "modelling" partners due to encountered computational difficulties. These were related to the stability of the numerical schemes in conjunction with the new models, but these problems were subsequently resolved. The other reason was that the two industrial partners use commercial codes in which the implementation of the new models can be made only by the code vendor, for which a high-level executive decision is required. AVL, itself a code vendor, has implemented some of the novel developments, whereas PSA uses exclusively marketed commercial codes of other vendors. Based on the models developed in this project, PSA may request their code vendors to implement at least some of the novelties.

Nevertheless, the knowledge and experience gained in the course of the project have provided valuable information about the performance of the models currently used in the CFD codes, their weaknesses and shortcomings, and on the potential and advantages of the new models developed in the project.

The experimental measurements performed both in a research and industrial laboratories, and data collected and processed, are a valuable addition to the relatively scarce data basis on vehicle aerodynamics and will most certainly be used in the future both by research institutions and automotive industries.

The developed unstructured LES solver is regarded as a valuable specific contribution to the simulation tools for the automotive industries. It is expected that this solver will be used in the future for optimal design of car mirrors, spoilers and other protruding elements. This tool will also be useful for providing input information (fluid velocity and pressure spectrum) for the computation of vehicle aerodynamic noise.

4. Exploitation Plans and Follow-up Actions

The outcome of the MOVA project: the turbulence models, experimental data base, LES code, and related findings and conclusions, will all provide an incentive both to the automotive industry, CFD vendors and research institutions active in automotive CFD, to improve their CFD software in term of accuracy, reliability and robustness, and to use it with more confidence in the design and development process.

All European transport automotive industries and research organizations are potential users of the knowledge and know-how generated in the project. The partners in the Consortium will stimulate the dissemination and the new know-how to the potential users. However, the primary users of the results will be the industrial partners in the Consoirtium. Specifically:

AVL as a CFD code vendor has already implemented the major findings on the new turbulence models in the latest versions of their SWIFT and FIRE codes, which have already been launched on the market. AVL also plans to implement some of the wall functions developed in the project, as well as other models to broaden the list of modeling options for customers. With these improvements, AVL will be able to offer the state-of-the-art in the CFD software, improving thus its market competitiveness. AVL intends also in the near future to offer to the market a new CFD code for LES for various specific purpose related to vehicle aerodynamics. Here the new unstructured solver, developed at

TU Delft, which has been to a great deal sponsored by AVL separately from the MOVA project, will serve as a basis.

PSA as a car manufacturer has been using extensively various commercial CFD software in the process of design and development of the automobile body shapes. In the past, the vehicle drag and lift were mainly in the focus of interest, but nowadays CFD is used more and more for other design purposes, such as predicting aerodynamic noise, reduction of force on windows, wheels interaction etc. The newly developed models offer a more reliable tool for these and other tasks. PSA needs to negotiate with its CFD providers in regard to implementing at least some of the project findings into their codes, for which executive decision is needed.

EDF-LNH has been in the past active in developing various multi-purpose CFD software and intended originally to implement the newly developed models into their N3S code. In meantime, EDF has been reorganized, the policy has changed and the CFD software development is not any more the prime focus of LNH activities. For that reason its not certain whether and in what extent EDF-LNH will exploit the project results.

The three university partners, TU Delft, UMIST and LSTM continue the activities in further development and examination of turbulence models and LES for various purposes. The experimental database generated in the project will be made available to a wider community as a basis for models calibration, verification and validation. The LSTM experimental results have already been made accessible to general public trough the ERCOFTAC database (European Research Community for Flow, Turbulence and Combustion), and served already as a database for one of the test cases in the 9th Workshop on Refined Flow Modelling, Darmstadt 4-5 October 2001. The same database will be used in the revisited exercise in the next 10th Workshop to be held in Poitiers, France, in 10-11 October 2002.

Publications

1. B. Basara (1999). Numerical simulation of turbulent wakes around a vehicle. *ASME Fluids Engineering DivisionSummer Meeting*, Proc.FEDSM99-7324, San Francisco, USA, July
2. B. Basara (2000). Computations of automotive flows using the second-moment closure. *European Congress on Computational Methods in Applied Sciences and Engineering, ECCOMAS*, Barcelona
3. Basara, A. Alajbegovic (1998). Steady state calculations of turbulent flow around Morel body. *7th Int. Symp. on Flow Modelling and Turbulence*, Taiwan
4. B. Basara and S. Jakirlic (2001). A new, hybrid modeling strategy for industrial CFD.
5. *Submitted for publication*
6. B. Basara, S. Jakirlic and V. Przulj (2001). Vortex-shedding flows computed using a new, hybrid turbulence model. S*ubmitted for publication*
7. B. Basara, V. Przulj and P. Tibaut (2001). On the calculation of external aerodynamics: industrial benchmarks. *SAE Conf.*2001-01-0701
8. S. Becker, H. Lienhart, C. Stoots (2000). Flow and turbulence structures in the wake of simplified car model (Ahmed model). *DGLR Fach Symp. AG STAB* (Stuttgard University), pp 15-17
9. T. J. Craft, H. Iacovides, and J. H. Yoon (1999). Progress in the use of non-linear two-equation models in the computation of convective heat-transfer in impinging and separated flows. *Flow, Turbulence and Combustion*, 63:59--80

10. T. J. Craft, A. V. Gerasimov, H. Iacovides, and B. E. Launder (2002). Progress in the generalization of wall-function treatments. *Int. J. Heat and Fluid Flow*, 23:148--160
11. S. E. Gant (1999). Assessment of wall function practices for impinging flows. *First-Year PhD Report,* Dept. of Mechanical Engineering, UMIST, Manchester, UK
12. Hadzic (1999). Second-Moment Closure Modelling of Transitional and Unsteady Turbulent Flows. *PhD thesis,* Faculty of Applied Sciences, TUDelft
13. K. Hanjalic (1999). Second Moment Turbulence Closures for CFD: Needs and Prospects. Int. J. CFD, Vol. 12, pp. 67 -97
14. K. Hanjalic and S. Jakirlic (2002). A new approach to modelling near-wall turbulence energy and stress dissipation. *J.Fluid Mech*, vol.459, pp.139-166
15. K. Hanjalic and S. Jakirlic (1998). Contribution towards the second-moment closure modeling of separating turbulent flows. *Computers and Fluids*, Vol. 27, No 2, pp 137-156.
16. K. Hanjalic and S. Jakirlic (2002). Second-moment turbulence closure modelling. *Closure Startegies for Turbulent and Transitional Flows*, Eds. B.E. Launder and N.D. Sandham, Cambridge University Press, pp 47-101.
17. R. Manceau and K. Hanjalic (2000). A new form of the elliptic relaxation equation for wall effects in RANS modelling. *Phys. Fluids*, Vol. 12, No 9, pp 2345-2351.
18. R. Manceau and K. Hanjalic (2002). A new near-wall Reynolds-stress turbulence closure. *Phys. Fluids*, Vol. 14, No 2, pp 744-754.
19. B. Niceno (2001). An unstructured parallel algorithm for large eddy and conjugate heat transfer simulations. *PhD thesis*, Faculty of Applied Sciences, TUDelft
20. B. Niceno and K. Hanjalic (2001). An Unstructured Finite-Volume Solver for Large Eddy Simulations. *submitted for publication*
21. C. M. E. Robinson (2001). Advanced CFD modelling of road-vehicle aerodynamics. *PhD thesis*, Dept. of Mechanical Engineering, UMIST