Perzon S. and Davidson L., On Transient Modeling of the Flow Around Vehicles Using the Reynolds Equations, In ACFD 2000 Beijing, Oct 17-20 2000, China, Eds. Wu J.-H., Zhu Z.-J., Jia F.-P., Wen X.-B. and Hu W., pp 720-727, 2000

ON TRANSIENT MODELING OF THE FLOW AROUND VEHICLES USING THE REYNOLDS EQUATION

PERZON Sven
Dept. 98365, CFD
Volvo Car Corporation
405 08 Göteborg
Sweden
email: sven.perzon@hq.vcc.volvo.se

DAVIDSON Lars
Dept of Thermo- and Fluiddynamics
Chalmers University of Technology
412 96 Göteborg
Sweden
email: lada@tfd.chalmers.se

Abstract

Transient simulations of the flow around the ASMO model from Daimler Benz have been done. The model is a well known geometry that is available for the public on internet. The results from the transient simulations are compared with experiments obtained from the Volvo model scale windtunnel and from Daimler Benz. Furthermore comparisons are made with stationary simulations. Transient simulations are of course very expensive and it is a fact that a transient simulation takes at least one order of magnitude longer to converge than a stationary simulation would do on the same mesh. However, a stationary simulation will not always give a stabile solution due to high resolution meshes and well behaved turbulence models that will remind us about the unstationary flow that exists in this type of engineering problems.

Introduction

Today CFD is used in the development of new trucks and cars at Volvo. Typically a new car or truck design is evaluated within 5 – 10 days after receiving a CAD database. Block-structured meshes of 2 – 4 million cells are generated with ICEM CFD. The equations are solved with the StarCD flow solver from Computational Dynamics Ltd. CFD is used to compare different design proposals, to do basic shape studies in the early stages of a project and to do shape optimization in later stages. The method predicts trends in drag with high accuracy in most cases but sometimes fails to do so. If CFD should be used on its own without model tests, the accuracy requirement increases and drag and lift must be truthfully simulated in the future.

In previous work by Ramnefors et. al. (1996), the base pressure error was believed to originate from an erroneous prediction of where the separation line was located. This is a fact if the geometry does not have a separation line that is well defined. However, studies of the flow around a simplified truck with a sharp trailing edge showed that even though the separation is geometrically well defined, similar error levels in the base pressure occur, see Perzon, Sjögren & Jönson (1998) and Perzon, Janson & Höglin (1999). This indicates that an erroneous predicting of the wake structure causes the error.

The mesh has been one of the major issues in all work concerning accuracy in drag predictions at Volvo. It is shown by Ramnefors et. al. (1996) and Axelsson et. al. (1998) that mesh quality may be just as important as mesh resolution when high accuracy is aimed for.

Different turbulence models have been used for stationary time-averaged flow. Some turbulence models mimic the wake structure well and others do not. Reynolds Stress Models have been used in the simulation of the flow around the Volvo ECC, (Ramnefors et al. 1996). It was shown that the qualitative pressure distribution was more accurately described but the base pressure level is poorly predicted. However, these complex models are often numerically troublesome, Perzon (1997). Hence, the route has been taken towards non-linear eddy viscosity models and algebraic Reynolds stress models see Perzon, Sjögren & Jönson (1998).

Numerical problems often occur when the mesh resolution becomes very fine around regions where a separation is present, e.g. the trailing edge. The separation line tends to move around when the solution is sought for during iterations and a fully converged solution may not be possible to find. If the geometry is complex enough so that poor mesh quality is unavoidable and/or if a complex turbulence model is used, the convergence problem will get even worse. It is often a fact that the wake is not stationary in reality and the attempt to solve the problem based on that assumption, cease to work at some modeling point. E.g. when the model mimics the flow well enough to discover this fact. The obvious solution to the convergence problem is to solve the equations using a transient solution methodology and that is what the present work will describe.

Transient simulations around vehicles have been presented previously by Summa (1992) and it was used for testing the vehicle for reactions of a pitch oscillation. The simulation was for demonstrational purposes and it shows an example of how transient simulations may be applied in a car project. Another area of interest is aerodynamical forces of a vehicle in cross wind gusts. Within that area only measurements have been presented previously, see for instance Garry et. al. (1994). This would be an excellent topic in the future for transient CFD. It would also be applicable to problems with passenger comfort in the tail coach of a train, which is known to be affected so that it vibrates slowly sideways by the time dependent side force caused by the unsteady wake. The latter is closely related to the current topic.

Tsuboi et. al. (1988) and Kataoka et. al. (1991) presented early transient simulations using quasi-direct numerical simulations, DNS. It is a fact that for such high Reynolds number, there is not a computer for sale today that could solve a true DNS of the flow around a car. Hence, when the transient Navier Stokes equations are solved without any modeling, it should be called DNS but since the resolution is way too coarse, it is hereafter refered to as, quasi-DNS. In Tsuboi et. al. (1988) the flow around a body with a slanted rear was simulated and the critical angle could be detected in the simulations. In Kataoka et. al. (1991), comparisons with forces and surface pressures from experiments agreed very well even for add on detailing. Yamada and Ito (1993) made a comparison between a transient quasi-DNS, and a stationary simulation using the k-ε model. It was shown that the stationary model was able to predict the flow field qualitatively correct but that it failed with the simulation of accurate forces. The quasi-DNS method simulated the surface pressure distribution poorly but predicted the forces fairly accurate. No data was presented on the unsteady behavior of the wake. The fact that forces may be accurately simulated for all the wrong reasons is something that is fairly common, see Perzon et. al. (1998). Furthermore, Horinouchi et. al. (1995), presented some results from a quasi DNS simulation where embedded grids was used. Also in this case, only stationary data was presented and the method was able to produce very accurate trends on aerodynamical forces. Aoki et. al. (1993) presented simulations on some test cases using a large eddy simulation, LES. The trends in drag were fairly accurately predicted. The critical roof slope is found in the simulations and some transient data are shown. LES are still a very expensive but interesting approach in vehicle aerodynamics. Murakami (1998) have done an excellent review of the capabilities of different LES models on flow over bluff bodies. Perzon and Davidson (2000) recently did a comparison between a transient RANS model and a simple LES model on the flow over a surface mounted cube.

Duell and George (1992) did transient measurements on a three dimensional bluff body in proximity to a ground plane with and without a splitter plate in the near wake. The measurement indicates that the aerodynamic drag is coupled to transient effects in the wake.

Unsteady pressure measurements in the wake have been done recently by Ishihara and Takagi (1999) where total pressure was measured at frequencies up to 300 Hz with and without moving belt. Comparisons were made between mean and instantaneous data and the difference were large implying that the wake large-scale movement is considerable for that vehicle.

Computational setup

The model chosen for this work is the well-known ASMO model, see figure 1. The geometry data can be downloaded form the World User Association, WUA, web page. The name, ASMO, is an abbreviation for "Aerodynamishes Studien Modell". This model was created approximately ten years ago in the Daimler Benz research department mainly for two reasons: to investigate if a body with very low drag could be created; and to have a neutral body not related to the development of an actual Mercedes car generation for testing CFD-codes. The characteristics of the body are: square back rear, smooth surface, boat tailing, underbody diffuser, no pressure induced boundary layer separation. The shape of the model is such that it can easily be meshed without the need for large computer resources. The virtual model were setup as close to the physical model as possible and the physical model was made according to,

- Frontal area: 0.062 m^2
- Wheel base: 0.540 m,
- Track: 0.220 m,
- Length: 0.810m,
- Color: Black

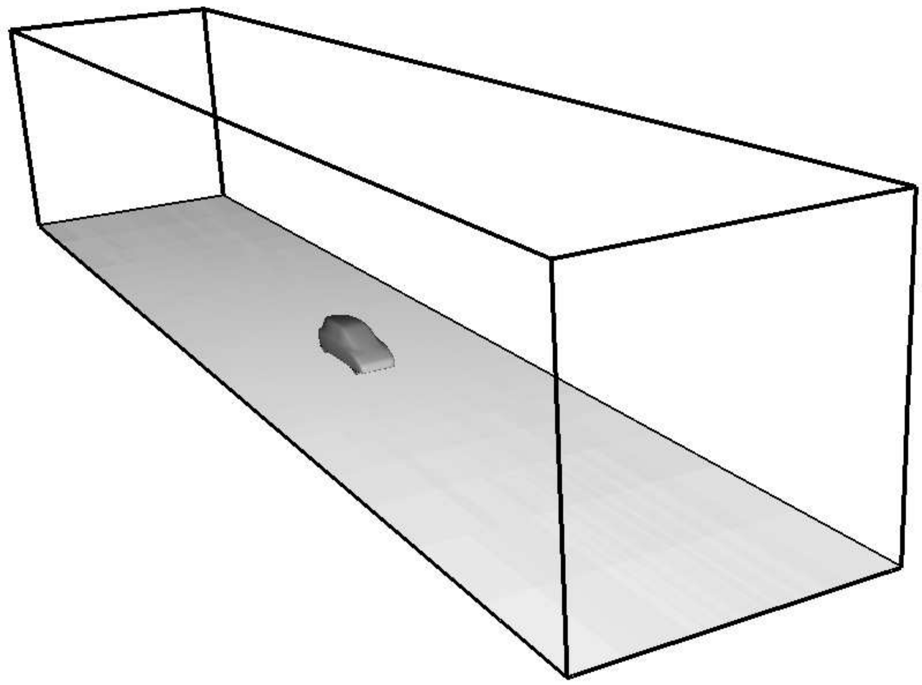


Figure 1. Computational domain. The ASMO model.

The experimental blockage ratio was 6 % based on the cross section in the Volvo model windtunnel which is a slotted wall windtunnel. However the computational domain describes a virtual 5th scale solid wall wind tunnel. The main characteristics are 3.67 m^2 cross section which gives a blockage ratio of 1.7%. The tunnel length is 9.0m of which 4m is located upstream of the model. Slip boundary conditions is applied to the tunnel roof, the tunnel side and to the entrance section of the tunnel floor. The car body and the rest of the tunnelfloor is treated as a normal wall with no slip boundary condition. Inlet boundary conditions were for velocity V_{in}=50 [m/s] and 0.1 % inlet turbulence intensity.

Slotted walls are more or less impossible to simulate with a reasonable amount of cells. Since the slotted wall windtunnel is designed for decreasing the effect of a high blockage ratio, it is comparable to a solid wall windtunnel with a lower blockage ratio and previous studies at Volvo have shown that the choise of the solid wall windtunnel described above is fairly accurate. The computational domain is shown in figure 1.

The mesh used for the simulations was a block structured hexahedral grid. For stationary models, only half the model is simulated using a symmetry boundary condition and the mesh size is around 380.000 hexahedral cells. The transient simulations have been done using a twice as large mesh, e.g. 760.000 cells, since the model is mirrored around the symmetry plane. The solver that has been used is StarCD and advection is discretized using the QUICK scheme for momentum and a first order upwind scheme for the turbulence quantities. The time descretization scheme is the second order accurate Crank-Nicholson scheme and the time step was constant, Δt=0.0005 [s]. It corresponds to a maximum Courant number of around 5.

Three turbulence models have been tested on the stationary simulations and they are the standard k-ε model of Launder and Spalding (1974), the nonlinear quadratic k-ε model of Shi, Zhu and Lumley (1993), the SZL-model, and the nonlinear cubic k-ε model of Lien, Chen and Leschziner (1996), the LCL-model. Wall functions were used with all models. In the transient simulations, the standard k-ε model and the SZL-model were used.

Results

Instantaneous velocity vectors of the flow behind the rear wheel segments in the diffuser region are shown in figure 2. It is shown that the flow is highly unsteady in this region and the wake from the wheel segments spreads towards the symmetry plane occasionally and periodically which indicates probable cause for dirt problems on the rear screen. Since this phenomenon occurs only sporadicly, a time averaged result would not give the proper result.

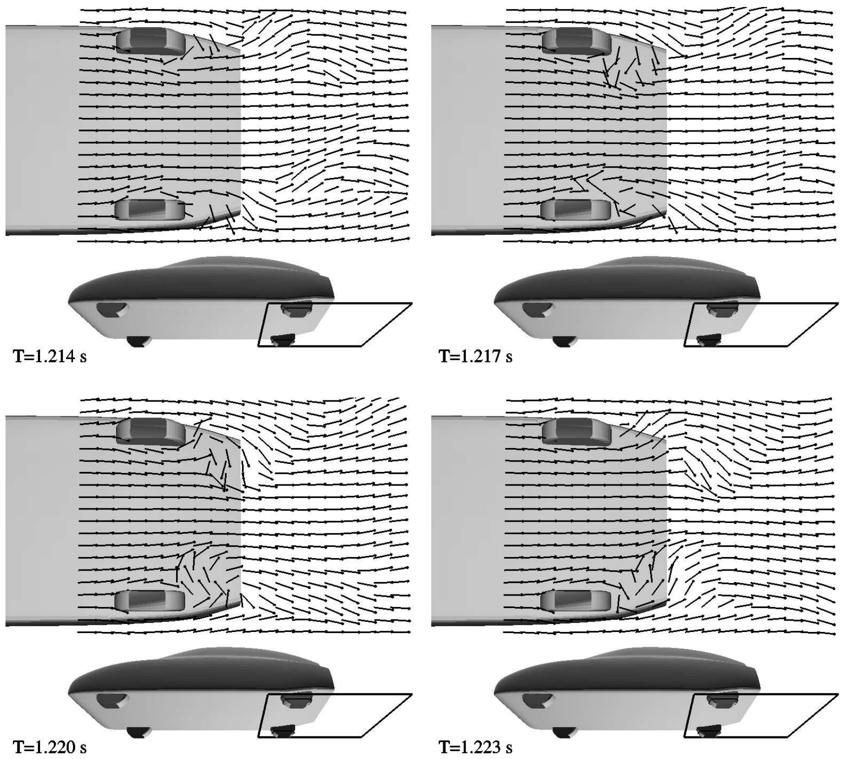


Figure 2. Velocity vectors at instants in time. The plane is located 15 mm above the tunnel floor which around half the ride height, (33.5 mm), for the vehicle. (The transient SZL-model)

The oscillating wheel wake seems to be the reason for a vertical vortex formation that is formed periodically at the trailing edge of the diffuser and sheds off when the wheel wake reaches the symmetry plane. The vortex formation have been noted previously by Nölting *et. al.* (1997).

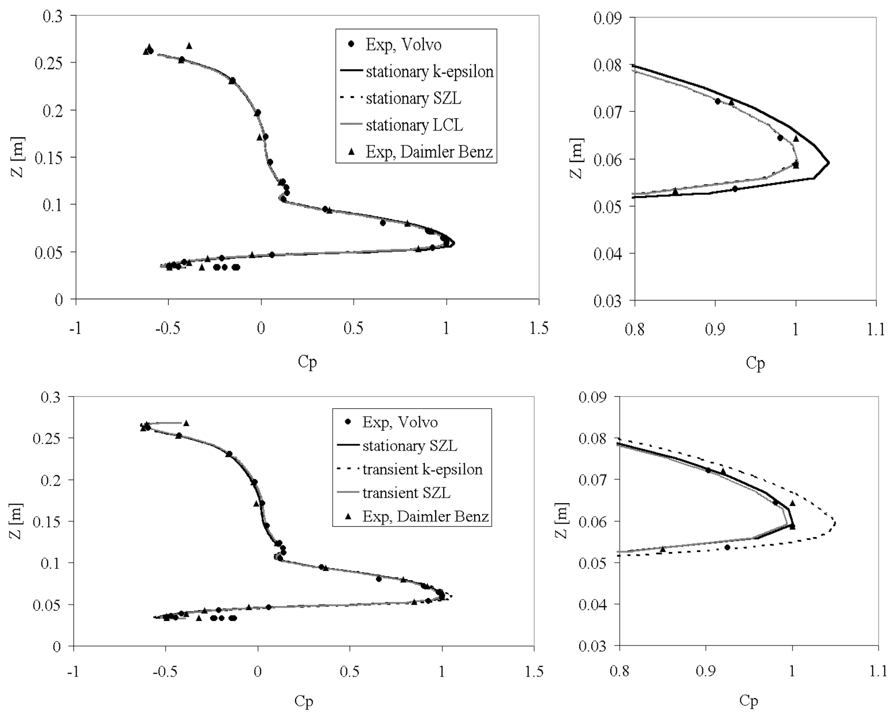


Figure 3. Pressure distribution for the different models over the front region in the symmetry plane. It is plotted against the vertical Z-coordinate. Note the stagnation point which is zoomed in to the right.

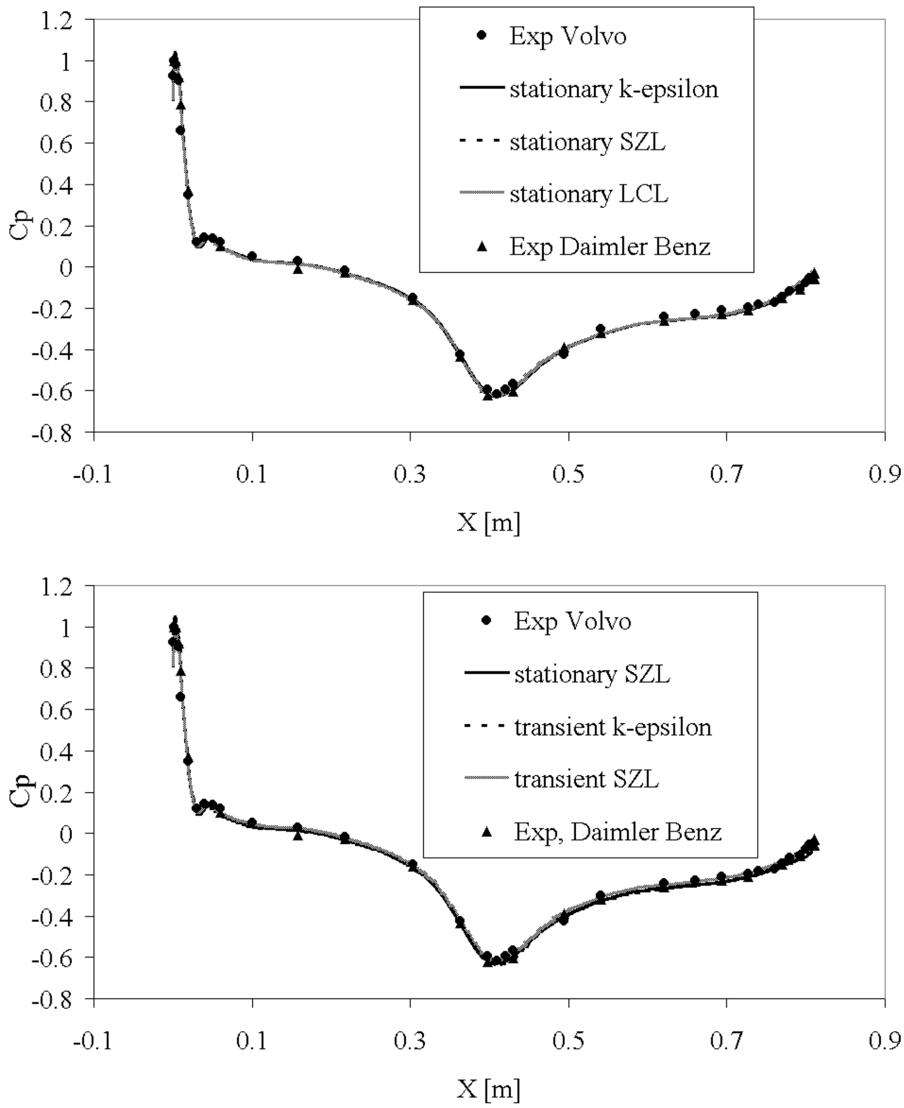


Figure 4. Pressure distribution over the roof along the symmetry plane on the ASMO using different turbulence models, stationary simulations (top) and transient simulations (bottom). The X coordinate is in the flow direction and the car spans from the front at X=0 to the rear screen at X=0.81 [m].

The pressure distribution along the front on the symmetry plane is shown in figure 3 for all cases. All models performs well except for the k-ε model which predicts the stagnation pressure too high which is true for both the transient and the stationary model. The transient SZL model is not more accurate than the stationary model in this region. Figure 4 shows the pressure distribution along the hood, over the front screen and over the roof of the vehicle in the symmetry plane. All models performs surprisingly well and the only clear difference between the them is in the latter part of the pressure recovery region close to the trailing edge. It is however the most important area since the boundary layer growth in this region sets the prerequisite for the base pressure level in the upper part of the body. The base pressure is shown in figure 5 and it is overpredicted by many of the models. The proper level of the base pressure for the simulated windtunnel is not known for sure as one can see from the deviation between the two experiments. It is however probably a fact that the Volvo model scale wind tunnel is more close to the simulated wind tunnel in terms of blockage, boundary layer thickness on the tunnel floor below the vehicle etc. The transient models seems to have lower base pressure than the stationary models. All stationary models and the transient k-ε model shows a higher pressure locally in the center of the base indicating that there is a local stagnation point at around Z=0.12, see figure 5. The experiments indicates that this is not a fact. The transient SZL model seems to mimic the wake structure correctly in this sence. Note that there is a big difference between the transient and the stationary simulation based on the SZL model. Furthermore, the base pressure level seems to be fairly well predicted when the transient SZL model is used.

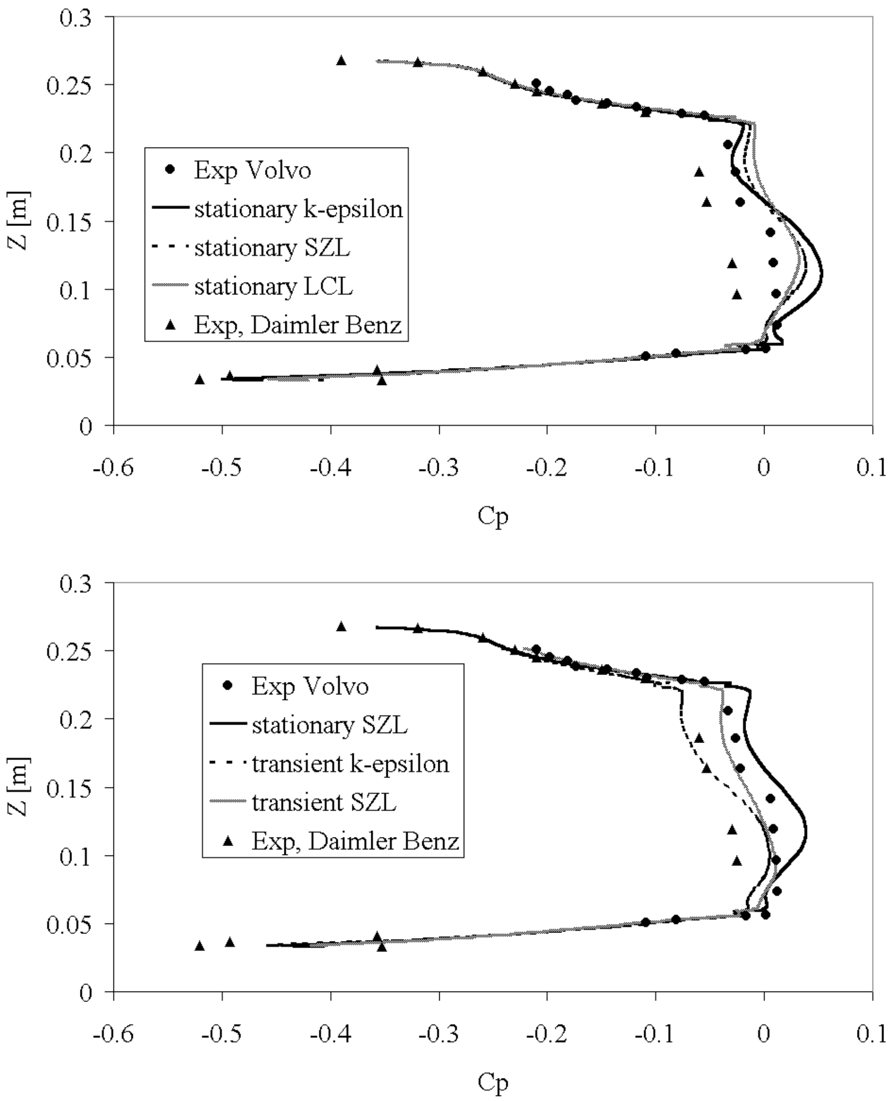
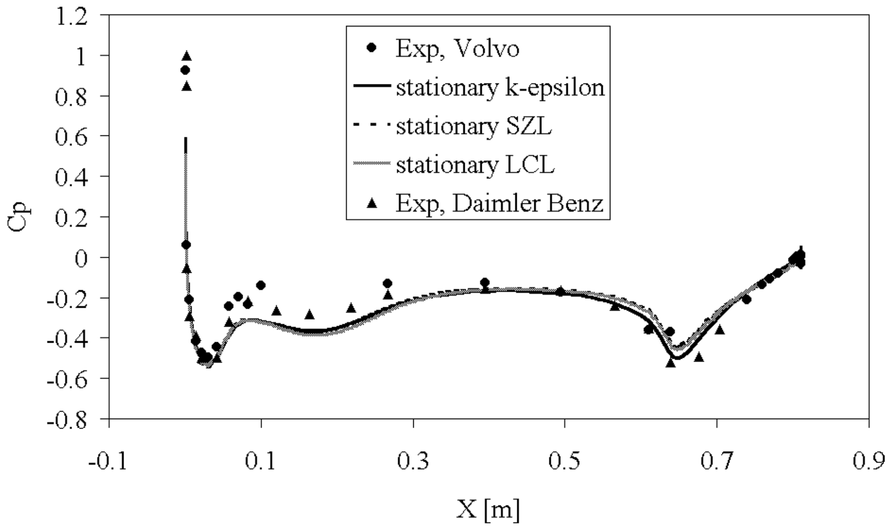


Figure 5. Base pressure distribution along the symmetry plane for stationary models, (top), and transient models, (bottom).

The pressure distribution along the underbody is shown in figure 6 and none of the models are able to capture the fairly steep pressure rize after the front radius. There is a very large difference between the two experiments in this region which indicates that the model is very sensitive to the prerequisites in this region. A very small deviation in the approaching velocity profile, a tiny error when mounting the model or a very small geometry defect may be the reason for the discrepancy between the experiments. Calculated velocity profiles immediately upstream of the car close to the wind tunnel floor corresponds well with PIV data of ditto which rules out that as a possible cause. Furthermore the model has been mounted several times in order to double check the data and no errors were found. When the model was intentionally mounted erroneous, it was found that the model is extremely sensitive to angle of attack. The ASMO geometry have been measured at Volvo and the result of this will be compared to the CAD database used for creating the mesh. Unfortunately, the measurement of the model is not fully completed yet and the comparison will thus not be included in this work.

The drag force have been calculated for all models and the result is shown in table 1. The deviation from experiments are significant for all models. The k-ε models, transient or not, consistently predicts the drag very high. The main reason for this is the stagnation pressure which is simulated too large.



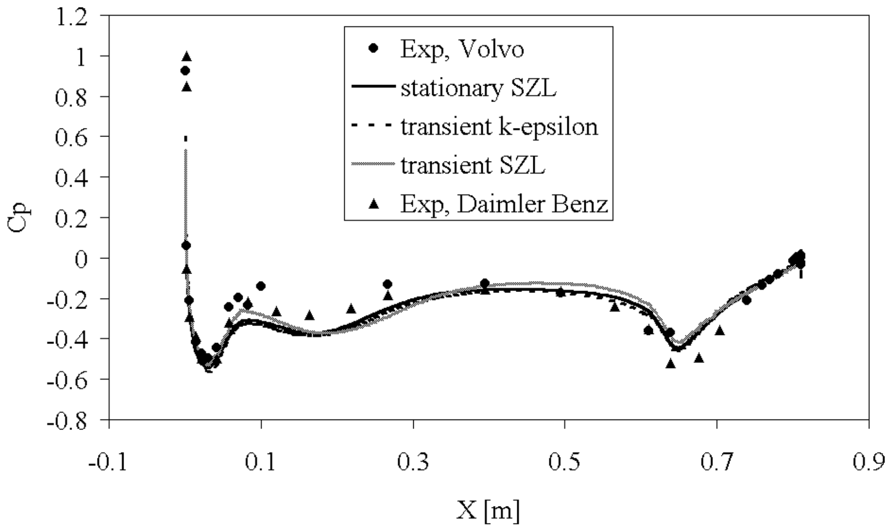


Figure 6. Pressure distribution along the car underbody along the symmetry plane on the ASMO using different turbulence models, stationary simulations (top) and transient simulations (bottom).

Model	Drag, C_d, (time averaged for the transient models)
Experiments, Volvo	0.158
Experiments, Daimler Benz	0.153
Stationary k-ε model	0.185
Stationary SZL-model	0.171
Stationary LCL-model	0.169
Transient k-ε model	0.195
Transient SZL-model	0.176

Table 1. Calculated forces for different models

Conclusions

Transient simulations based on the Reynolds equations using a turbulence model can be used within vehicle aerodynamics. The computation time increases by at least one order of magnitude. However, there is always a solution. More elaborate stress-strain relationships such as the SZL model have been used with success and the wake structure is simulated well with this model while the corresponding stationary model did not show this benefits. Future work will investigate the poor performance in the underbody region.

Acknowledgment

I would like to thank Volvo Car Corporation and the foundation for Integral Vehicle Structures for funding this work. I also would like to send my gratitude to Dag Aronson who conducted all the experiments.

References

AOKI K., OHBAYASHI T., ZHU M. and MIYATA H., 1993, Finite-volume simulation of 3-D vortical flow-fields about road vehicles with various after-body configuration , SAE technical paper 931896

AXELSSON, N., RAMNEFORS, M. and GUSTAFSSON, R., 1998 Accuracy in Computational Aerodynamics, Part 1: Stagnation Pressure, SAE Technical Paper 980037.

DUELL E. G. and GEORGE A. R., 1992, Unsteady wakes of three dimensional bodies, In *Bluff-body wakes, dynamics and instabilities*, Eds. Eckelmann et. al., Springer-Verlag, pp. 293-296

GARRY K. P., MACKLIN A. R. and van OPSTAL E. P. E., 1994, Measurement of transient aerodynamic loads on bluff bodies at extreme yaw angles, In *Vehicle aerodynamics*, The Royal Aeronautical society, Loughborough Univ. Of Tech., UK, July 18-19

HORINOUCHI N., KATO Y., SHINANO S., KONDOH T. and TAGAYASHI Y., 1995, Numerical investigation of vehicle aerodynamics with overlaid grid system, , SAE technical paper 950628

ISHIHARA Y. and TAKAGI M., 1999, Unsteady pressure analysis of the wake flow behind a passanger car model, SAE technical paper 990810

KATAOKA T., CHINA H., NAKAGAVA K., YANAGIMOTO K. and YOSHIDA M., 1991, Numerical simulation of road vehicle aerodynamics and effect of aerodynamic devices, SAE technical paper 910597

LAUNDER, B. E. and SPALDING, D.B., 1974, The numerical computation of turbulent flow , *Comp. Meth. In Appl. Mech. and Eng.* 3, p. 269

LIEN F.S., CHEN W.L. and LESCHZINER M.A., 1996, Low-Reynolds-number-viscosity modelling based on non-linear stress-strain/vorticity relations, Proc. *3rd Symp. on Engineering Turbulence Modelling and Measurements* , Creete, Greece

MURAKAMI S., 1998, Overview of turbulence models applied in CWE-1997, *J. of Wind Eng. and Industrial Aerodynamics*, 74-76, pp. 1-24

NÖLTING S., ALABEGOVIC A., ANAGNOST A., KRUMENAKER T. and WESSELS M., 1997, Validation of Digital Physics simulations of the flow over the ASMO-II body, Internal report 97VR092, EXA Corp.

PERZON, S., 1997, Reynolds Stress Modeling of Flow Separation on Curved Surfaces , Thesis for the degree of Licenciate of Engineering, Chalmers University of Technology, Göteborg, Sweden

PERZON, S., SJÖGREN, T., and JÖNSON, A., 1998, Accuracy in Computational Aerodynamics, Part 2: Base pressure , SAE Technical Paper 980038.

PERZON S., JANSON J. and HÖGLIN L., 1999, On comparisons between CFD methods and wind tunnel tests on a bluff body, SAE technical paper 990805

PERZON S. and DAVIDSON L., 2000, On CFD and transient flow in vehicle aerodynamics, SAE Technical Paper 2000-01-0873

RAMNEFORS, M., BENSRYD, R., HOLMBERG, E., and PERZON, S., 1996, Accuracy of Drag Predictions on Cars Using CFD - Effect of Grid Refinement and Turbulence Models , SAE Technical Paper 960681.

SHIH, T. H., ZHU, J. and LUMLEY, J. L., 1993, A realizable Reynolds stress algebraic equation model , NASA tech. Memo. 105993

SUMMA J. M., 1992, Steady and unsteady computational aerodynamics simulations of the Corvette ZR-1, SAE technical paper 921092

TSUBOI K., SHIRAYAMA S. OANA M. and KUWAHARA K., 1988, Computational study of the effect of base slant, In *2nd International Conference on supercomputing in the automotive industry*, Seville Spain

YAMADA A. and ITO S., 1993, Computational analysis of flow around a simplified vehicle-like body, , SAE technical paper 930293