Perzon, S. and Davidson L., On CFD and transient flow in vehicle aerodynamics, SAE Technical paper 2000-01-0873, 2000

2000-01-0873

On CFD and transient flow in vehicle aerodynamics

Sven Perzon
Volvo Car Corporation

Lars Davidson
Chalmers University of Technology

ABSTRACT

The accuracy of computational fluid dynamics, CFD, has improved considerably over the years but still, large errors are present and vehicle parameters such as drag and lift are often poorly predicted.

The current work is investigating how transient CFD would cope with a very complex flow structure around a surface mounted cube. A transient Reynolds averaged Navier Stokes model, RANS model, is presented together with a large eddy simulation model, LES model.Furthermore, two “industrial like” test cases have been simulated using a transient RANS model.

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 a very high accuracy in most cases but sometimes still 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). This indicates that an erroneous predicting of the wake structure causes the error.

The mesh has been one of the major issues in all the accuracy work done 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 accuracy is searched for.

Different turbulence models have been used for the 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 and the attempt to solve the problem based on that assumption, cease to work at some modeling point. E.g. when the accuracy in the model is large enough to discover this fact. The obvious solution to the convergence problem is to solve the equations using a transient solution methodology and this 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 Garry et. al. (1994) and the work by Ryan and Dominy (1998). This would be an excellent topic in the future for transient CFD. It should be noted also that Hucho and Emmelmann (1974) have proposed a theoretical prediction method.

Another area where transient CFD would be applicable is 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 transient 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, I have preferred to call it, 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 transient 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 subgrid scale model presented was a Smagorinsky (1965) model and the trends in drag were fairly accurately predicted. The critical roof slope is found in the simulations and some transient data is shown.

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.

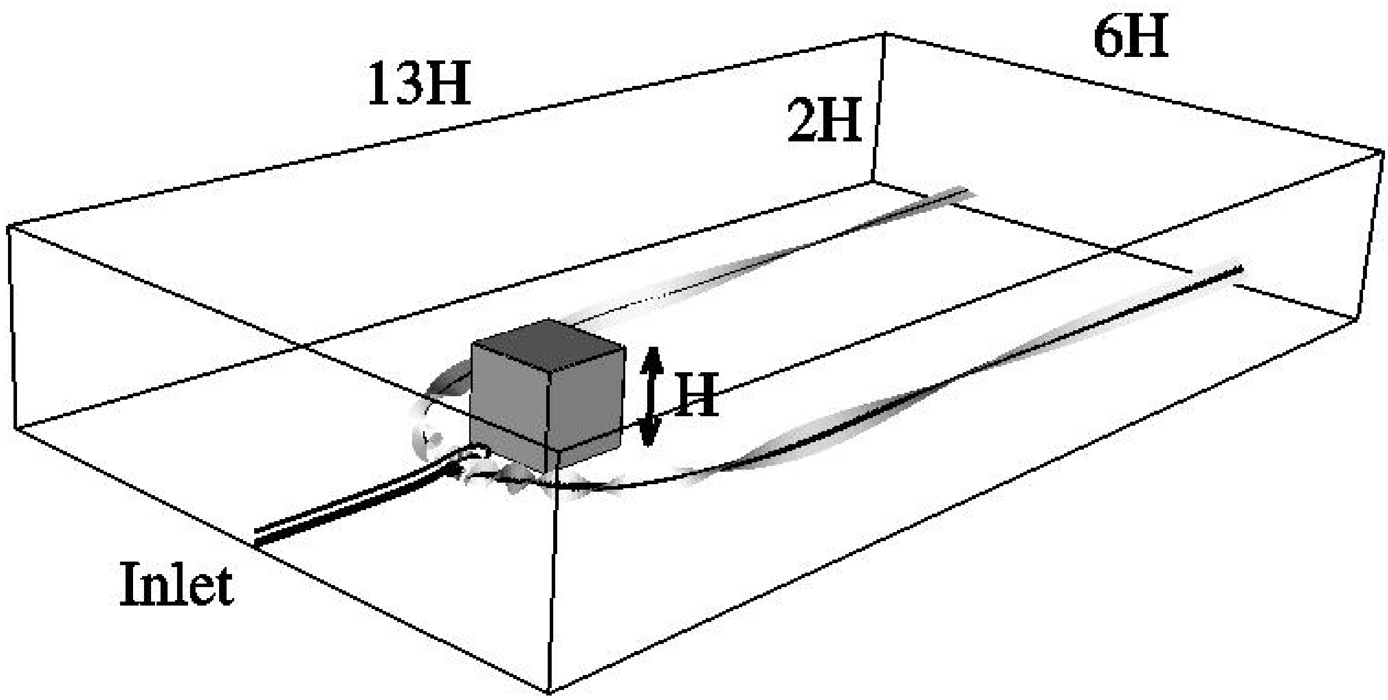

Figure 1 Computational domain and visualisation of the horse shoe vortex with streamlines from the averaged velocity field of the LES simulation.

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.

SURFACE MOUNTED CUBE

The surface mounted cube has been measured by Martinuzzi and Tropea, 1993, and was used as the main test case for this study. Several others have studied it before, e.g. see Krajnovic and Davidson (1999). Furthermore it has been one of the test cases in the ERCOFTAC "classic collection" database, which means that the measurements have been checked by an independent academic community which give reason to believe that it is measurements of high quality. Database adress:

http://vortex.mech.surrey.ac.uk/

THE COMPUTATIONAL SETUP

The computational domain was created with the ability of testing several models and schemes and hence, the model had to be kept fairly coarse.

<u>The mesh</u>

ICEM Mulcad was used for grid generation and the mesh that was used for the study had the following characteristics,

- Number of cells: 264576 hexahedral cells.
- Block structured grid
- The simple geometry made it possible to obtain perfect orthogonality everywhere in the mesh

The inlet of the domain is located 3 cube heights upstream of the cube and the overall length of the domain is 13 cube heights, see figure 1.

<u>Turbulence models</u>

In Hucho (1998), CFD methods are classified into several groups where the latter three groups are,

- Time averaged, steady, RANS methods
- Large eddy simulation, LES
- Direct numerical simulation, DNS

Of these groups the first stationary RANS is the most widely used and mostly in conjunction with the k-ε turbulence model. The reasons for using the k-ε model are that the model is numerically robust and that is has been used for quite some time, giving presence to an extensive validation database. The latter reason if of great importance in industrial applications. A tremendous amount of work would be required to build a similar knowledge about another model and hence the k-ε model is used even though it is known to give the wrong answer in many typical situations. Stagnation regions and regions with high streamline curvature are typical flows where the model fails. Due to the historic rucksack it is troublesome for industry to start using more elaborate turbulence models since many previous simulations have to be recomputed with the new model. A similar problem about passenger comfort within the aircraft industry is nicely pin pointed in Adams (1997).

As described below, RANS can be used in transient simulations. Since LES, is still too expensive in most cases, transient RANS may be an alternative for some years until computers are fast enough to allow for LES simulations.

As hinted previously, two different classes of turbulence models have been used during the course of this study and these are a time averaging method, RANS, and a method that is filtering the equations of state, LES.

The time averaging approach decomposes velocities and pressure into a mean and fluctuating part and the Navier Stokes equations are time averaged over some time interval, **T**, and the Reynolds equations appear,

$$\frac{DU_i}{Dt} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\nu\frac{\partial U_i}{\partial x_j} - \overline{u_i u_j}\right) \qquad (1)$$

A new term has to be taken care of by some kind of turbulence model. As stated above, the most common choice in industrial applications is the standard k–ε model, Jones and Launder (1972). In this study a more advanced two-equation model have been used and it is the quadratic stress-strain relationship proposed by Shih et. al. (1991), hereafter denoted the SZL-model. Even though a time averaging has been imposed to the equations, a transient simulation can be appropriate if the time averaging interval is long enough to embed the turbulent time scales and short enough to capture the large scale mean flow movements. In a flow with a moving boundary such as an in-cylinder flow it is easily understood that some scales are turbulent and that some scales originate from moving boundaries, e.g. piston and valves. In aerodynamics the boundaries are stationary and it may be harder to separate turbulence from mean flow variations. One may suspect that such a model would work properly only when a clear scale separation is present. An example of such a case would be an obvious vortex shedding behind a bluff body . Then the mean flow variations would be the moving boundary of the wake while the chaotic structure inside the wake would be turbulence. If the wake movement would have several characteristic frequencies and the scales spans from the largest scales to the smallest turbulent scales, it will be tricky to choose the time averaging period, **T,** that fulfilled the requirement described previously. The transient approach using a turbulence model is usually called T RANS, (Transient Reynolds Averaged Navier Stokes).

The other approach that has been used is a large eddy simulation, LES, strategy. Direct simulation of all scales in a turbulent flow is feasible only at very low Reynolds numbers. This suggests a simulation technique based on some decomposition of the flow into large and small-scale structures. The large scales are then directly solved for in three-dimensional time-dependent fashion and the small scales are modeled. This approach is called LES, (Large Eddy Simulation). The major reason for testing such a model in this work is that it should, by definition, work better in flows where there is a lack of scale separation since the model is designed to take care of all large transient scales, turbulent or not. The velocities and the pressure are decomposed into a resolved part (still transient) and a subgrid part, which is modeled. The result becomes similar to the Reynolds equations,

$$\frac{D\overline{u}_i}{Dt} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\nu \frac{\partial \overline{u}_i}{\partial x_j} - \tau_{ij}\right) \tag{2}$$

The last term have to be modeled and the most simple sub-grid model available is the Smagorinsky (1965) and it has been used in this study. The grid length scale was chosen as, $V^{1/3}$, where V is the grid cell volume. A nice overview of other interesting sub-grid models tested on flows around bluff bodies can be found in Murakami (1998).

The subgrid model in the LES model will vanish when the mesh is refined to an extent that all fine scales are resolved. Thus, the LES model becomes a direct numerical simulation when the mesh and time step allows for it. The turbulence model in the T RANS model will never fade out because of a fine mesh resolution.

Boundary conditions

The inlet boundary was taken from experiments, Martinuzzi and Tropea, 1993. This experimental database also provided data for the normal and shear stresses at the inlet and the turbulence properties k and ε were set, if appropriate, according to,

$$k = \frac{1}{3}\overline{u_i u_i} \tag{3}$$

$$\varepsilon = \frac{k^{\frac{3}{2}}}{C_\mu^{-0.75}\max(\kappa\eta, \lambda\delta)} \tag{4}$$

where η is the distance to the wall, δ is the boundary layer thickness at the inlet and λ=0.09 for channel flows.

At the walls the wall function approach was used on the tunnel roof, on the cube and on the tunnel floor. The sides of the domain were treated as slip walls, which also was used in the work of Krajnovic and Davidson (1999).

Numerical model

The commercial solver StarCD was used throughout this study and the version was 3.05. The RANS model is available in the code and the LES model was easily incorporated via user coding.

The Crank Nicholson scheme was used for the time discretization. Two different spatial discretization schemes were tested and these are the central differencing scheme, which is 2nd order accurate, and the MARS scheme, which also, is a 2nd order accurate scheme. Normally the central differencing, CD, scheme is not robust enough in stationary simulations but since the simulation is time dependent, it seems to be robust enough and a very accurate and " low cost" scheme to use. The MARS scheme with two different blending factors, 0.5 and 1.0, that controls the amount of upwinding in the scheme. Low values of this factor introduces more upwinding. The MARS scheme is second order accurate regardless of the blending factor.

The equations are solved for using the PISO algorithm. Different time steps have been tested for the transient RANS model but in all comparisons between the transient RANS and the LES model, an identical time step was used, **Δt = 0.01 s**, which corresponds to a maximum Courant number around 3.

RESULTS

The initial simulation that have to be performed prior to any data collection was set to ten time scales, τ. The time scale, τ, is the time it takes for a fluid particle in the free-stream to pass the computational domain, e.g. $\tau = 10$ **s**. Hence, with **Δt = 0.01 s** it takes 100.000 time steps before data collection can be started.

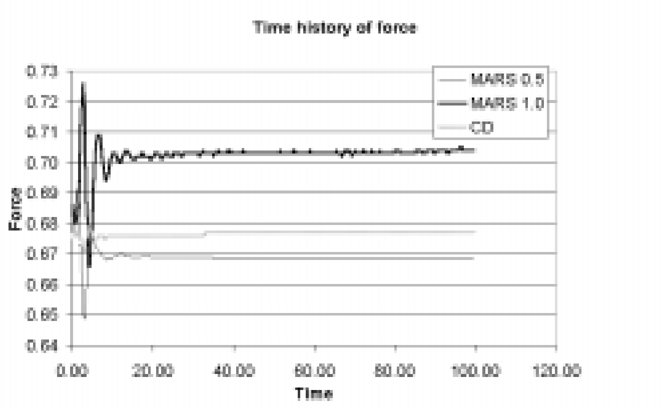

Figure 2. Time history of the force on the surface mounted cube during the initialization phase. A comparison between different advection schemes. $\Delta t = 0.1$ seconds.

The aerodynamical drag force of the cube is shown in figure 2 as a function of time during the initialization phase for the SZL model. The effects of the chosen scheme is shown and as can be seen the very simple CD scheme is similar to the much more elaborate and computationally expensive MARS scheme. In figure 3, a similar plot is showing the effect caused by different time steps using the SZL model. If the time step is chosen as too coarse, it may be a fact that transient behavior will be damped out. It is also shown that the drag seems to have just about settled and the initialization phase appears to be long enough. The averaging are done from the end of this initialization phase and the statistics was collected for another 10 time scales described above.

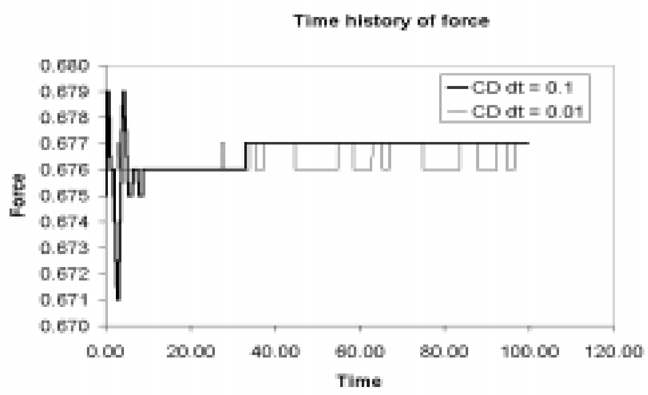

Figure 3. Time history of the force on the surface mounted cube during the initialization phase. A comparison between different time steps. $\Delta t = 0.1$ seconds corresponds to a maximum Courant number around 30 and $\Delta t = 0.01$ seconds coincide with ditto around 3.

In figure 4 comparisons are made between LES, transient RANS and experiments, 0.5 cube heights upstream of the front face on the cube, see figure 1. It is readily shown that the transient RANS model is fairly close to experiment. The experimental position goes through the horseshoe vortex and the RANS model mimics that quite well. It exaggerates the back-flow somewhat. The LES model seems to have a problem at Y/H=1.1 where an unphysical peak is shown when the nondiffusive central differencing scheme, CD scheme, is used. The same turbulence model does not show this when the MARS scheme is used. The MARS scheme and the CD scheme are both second order accurate but the MARS scheme contains some upwinding which in this case damps out the numerical instabilities that causes the peak whereas the CD scheme doesn't. It is interesting to note that the TRANS model does not show this behaviour even though the CD scheme is used. The turbulence model incorporates enough turbulent diffusion to damp out this phenomenon. The LES model exaggerates the backflow somewhat less than the TRANS model. Figure 5 shows the vertical velocity component at the same spot and the TRANS model is less poor in comparison with the LES model. However, none of the models predicts the profile well in this region. In experiments the horseshoe vortex has its center closer to the cube than the X location of the profile. Hence, the vertical component of the velocity becomes positive close to the wall. This is not a fact in any of the simulations where all models have a negative vertical velocity component close to the floor implying that the vortex is located on the other side of the measuring location.

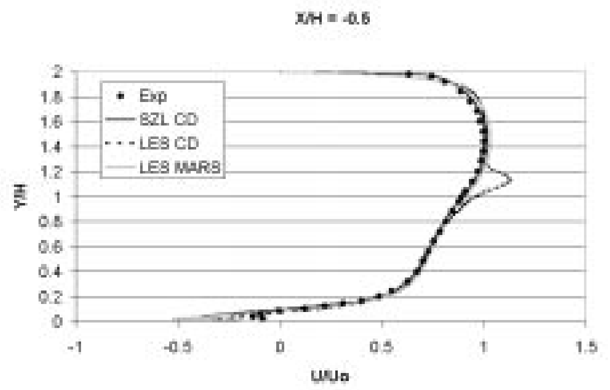

Figure 4. Streamwise velocity component, U, at 0.5 cube heights upstream of the cube front side, e.g. at X/H = -0.5. The data is taken along the vertical coordinate axis, Y, in the symmetry plane, Z = 0.

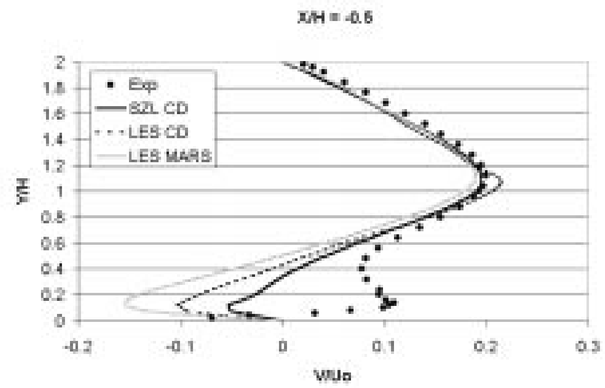

Figure 5. Vertical velocity component, V, at 0.5 cube heights upstream of the cube front side, e.g. at X/H = -0.5.

On the cube roof in figure 6 the flow from the front edge is shown from the cube roof to the channel roof. The TRANS model is more accurate than the LES model with the central differencing scheme and similar to the LES model with the MARS scheme. The next location, see figure 7, is on the trailing edge of the cube roof. The LES model with the CD scheme underpredicts the separation on the roof and the other two models overpredicts the region of separation slightly.

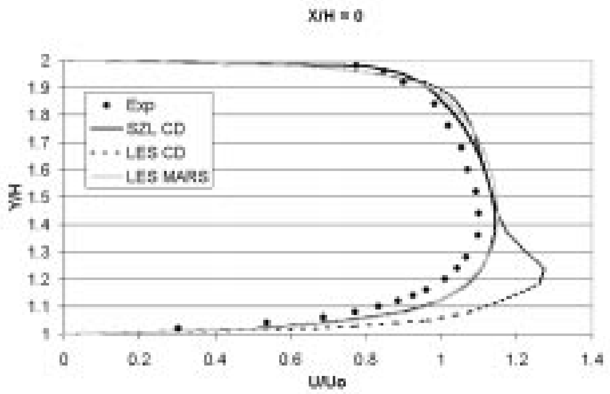

Figure 6. Streamwise velocity, U, profile on the symmetry plane and on the cube roof at
X/H = 0.

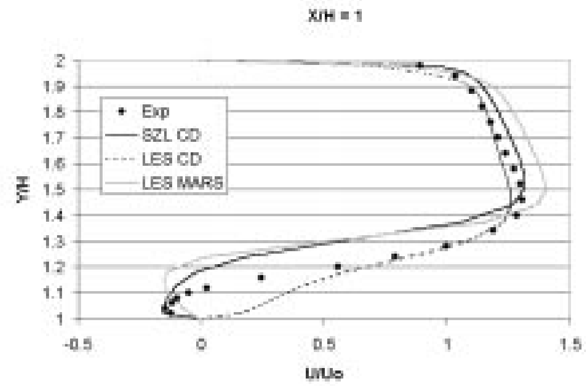

Figure 7. Streamwise velocity, U, profile on the symmetry plane and on the cube roof at the trailing edge, e.g. X/H = 1.

One cube height downstream, X/H = 2, the LES MARS and the TRANS models still overdo the separation which is a direct consequence of the upstream error, see figure 8. Next location is located at X/H = 6 and both the LES models closes the wake much more rapidly than the TRANS model and thus at this point, the LES models are closer to experiment then the TRANS model, see figure 9.

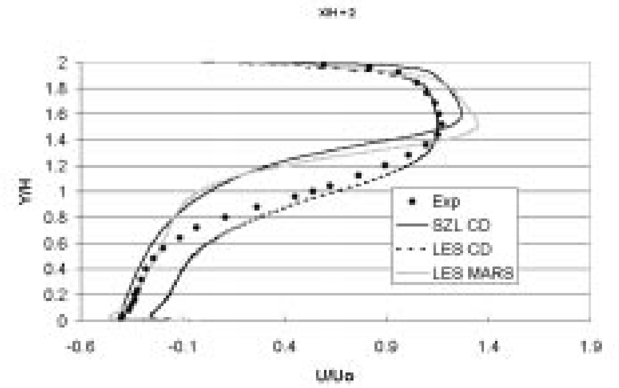

Figure 8. Streamwise velocity component, U, 1 cube height downstream of the cube trailing edge, e.g. at X/H = 2.

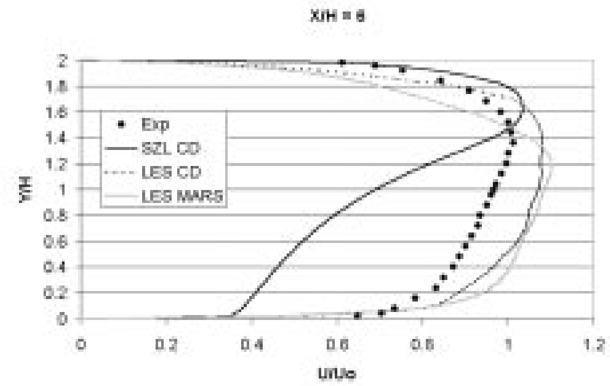

Figure 9. Streamwise velocity component, U, 5 cube height downstream of the cube trailing edge, e.g. at X/H = 6.

OTHER CONFIGURATIONS

The surface mounted cube is a relevant test case in the sense that it is a well-measured test case and it has a very complex flow structure. The fact that it is a very simple geometry makes it even more suitable for testing computational models. However, the Re number is very low, Re=80000, and it lacks underbody flow and hence, some other models must be tested in conjunction with the surface mounted cube. The Volvo ECC has been studied before by Ramnefors et. al. (1996). The model was simulated using the same quadratic k-ε model as was presented previously as the SZL-model.

The time history of the drag is shown in figure 10. The transient behavior settles to a nice and smooth sinusoidal curve. The magnitude of the oscillations is however very small. It indicates that the flow behind the Volvo ECC is very calm. No large vortex structures or other "bad" aerodynamical features exist on that model which is shown in the very low experimental drag figure, see table 1.

Configuration	C_d
Time averaged TRANS. SZL model.	0.115
Stationary RANS. SZL model.	0.111
Stationary RANS. k-ε model.	0.143
Experiments	0.141

Table 1. A comparison of drag figures between stationary and transient RANS simulations.

Stationary simulations on the identical mesh have been computed and all drag figures are tabulated in table 1. Note that the simple k-ε model computes a drag figure that is very close to the measured value. However, the reason for this is that the model overpredicts the stagnation pressure which cancel out the error in the base pressure. The base pressure error are of the same order for all turbulence models.

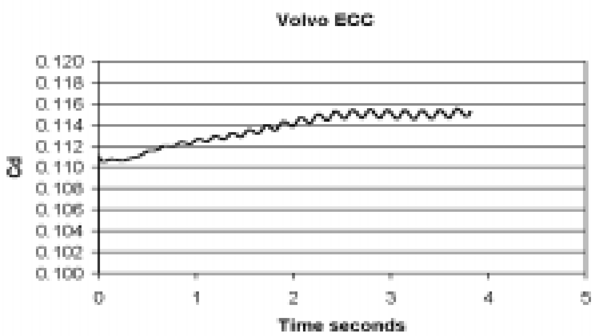

Figure 10. Time history of drag for the Volvo ECC.

A simplified Volvo FH truck have been simulated and this configuration have a very clear oscillating wake like a fish tail. The wake oscillates sideways in the simulation which is clearly shown in figure 11.

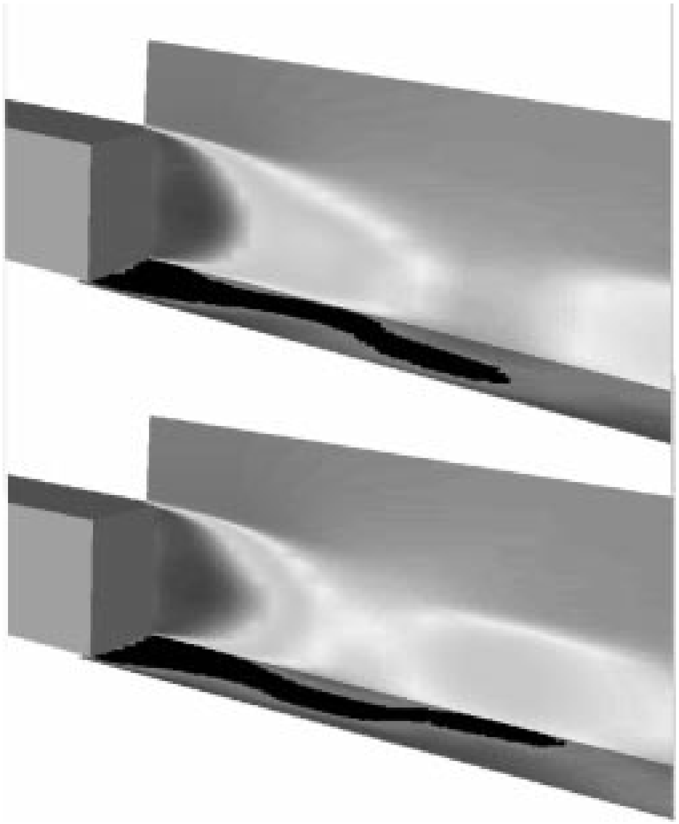

Figure 11. Velocity isocontours in the wake. Snapshots at two instants in time.

CONCLUSION

Simulation of transient flows is a fruitful path in future development of cars. The main benefits are,

- Stability, there is always a solution to the equations. Hence, it doesn't matter how transient the flow is, there is always a solution.
- Physics, by solving the transient equations, more physics are imported to the problem. There is one less assumption made.

The simulation times becomes larger, around 10 times larger but the enhanced stability makes it possible to use more advanced physical models which in turn may give higher accuracy with a coarser model.

CONTACT

Sven Perzon, Volvo Car Corporation, Dept. 98365, Bld PVIB3, 405 08 Göteborg, Sweden
Phone: (+46) 31 3213149
Fax: (+46) 31 593514
email: Sven.Perzon@hq.vcc.volvo.se

REFERENCES

1. Adams S., The Dilbert future. Thriving on stupidity in the 21st century., 1st edition, HarperCollins Publishers Inc., ISBN 0-88730-866-X, 1997
2. 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
3. Axelsson, N., Ramnefors, M. and Gustafson, R. (1998) Accuracy in Computational Aerodynamics, Part 1: Stagnation Pressure, SAE Technical Paper 980037.
4. 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, 1992 , pp. 293-296
5. 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, 1994
6. 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
7. Hucho W.-H. (1998), Aerodynamics of road vehicles, 4th edition, ISBN 0-7680-0029-7, SAE
8. Ishihara Y. and Takagi M. (1999), Unsteady pressure analysis of the wake flow behind a passanger car model, SAE technical paper 990810
9. Jones, W. P. and Launder, B. E. (1972) The prediction of laminarization with a two-equation model of turbulence , Int. J. of Heat and Mass Transfer 15, 310-314
10. Kataoka T., China H., Nakagawa K., Yanagimoto K. and Yoshida M. (1991), Numerical simulation of road vehicle aerodynamics and effect of aerodynamic devices, SAE technical paper 910597
11. Krajnovic S. and Davidson L. (1999) Large eddy simulation of the flow around a surface mounted cube using a one-equation subgrid model, In the first international symp. on turbulence and shear flow phenomena, Santa Barbara, Sept 12-15 1999
12. Martinuzzi R. And Tropea C., (1993), The Flow Around a Surface-Mounted, Prismatic Obstacles Placed in a Fully Developed Channel Flow, J. Fluids Eng. pp. 85-91, Vol. 115, March.
13. Murakami S. (1998) Overview of turbulence models applied in CWE-1997, J. of Wind Eng. and Industrial Aerodynamics, 74-76, pp. 1-24
14. 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
15. Perzon, S., Sjögren, T., and Jönson, A., (1998) Accuracy in Computational Aerodynamics, Part 2: Base pressure , SAE Technical Paper 980038.
16. 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
17. 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.
18. Shih, T. H., Zhu, J. and Lumley, J. L. (1993) A realizable Reynolds stress algebraic equation model , NASA tech. Memo. 105993
19. Smagorinsky J., Manabe S. and Halloway J. L. (1965), Numerical results from a nine-level general circulation model of the atmosphere, Monthly weather review, 93, Dec 1965, pp. 727-768
20. Summa J. M. (1992), Steady and unsteady computational aerodynamics simulations of the Corvette ZR-1, SAE technical paper 921092
21. 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
22. Yamada A. and Ito S. (1993), Computational analysis of flow around a simplified vehicle-like body, , SAE technical paper 930293