



OTC 18348

Application of CFD for Vortex-Induced Vibration Analysis of Marine Risers in Projects

Mark Dixon, Dave Charlesworth – DeepSea Engineering & Management

Copyright 2006, Offshore Technology Conference

This paper was prepared for presentation at the 2006 Offshore Technology Conference held in Houston, Texas, U.S.A., 1–4 May 2006.

This paper was selected for presentation by an OTC Program Committee following review of information contained in an abstract submitted by the author(s). Contents of the paper, as presented, have not been reviewed by the Offshore Technology Conference and are subject to correction by the author(s). The material, as presented, does not necessarily reflect any position of the Offshore Technology Conference, its officers, or members. Papers presented at OTC are subject to publication review by Sponsor Society Committees of the Offshore Technology Conference. Electronic reproduction, distribution, or storage of any part of this paper for commercial purposes without the written consent of the Offshore Technology Conference is prohibited. Permission to reproduce in print is restricted to an abstract of not more than 300 words; illustrations may not be copied. The abstract must contain conspicuous acknowledgment of where and by whom the paper was presented. Write Librarian, OTC, P.O. Box 833836, Richardson, TX 75083-3836, U.S.A., fax 01-972-952-9435.

Abstract

The application of CFD (Computational Fluid Dynamics) to the issue of vortex induced vibration (VIV) of marine risers is a rapidly developing area of engineering which offers the potential to improve design capability whilst reducing associated testing costs. However, the tools and processes to achieve this are still being developed, tested and validated by a number of groups within the offshore industry and as yet only sporadic project application has been made of CFD for VIV analysis of marine risers.

This paper discusses the key issues associated with the application of CFD to projects followed by a case study illustrating a suitable area of application given the current level of maturity of the numerical tools and the timeframes required to perform the work. The first part of the paper includes a review of the available turbulence models comparing their benefits and drawbacks as well as the important issue of validation and benchmarking with recommendation on acceptable levels of accuracy given the application objectives and requirements.

The case study is based on a deepwater drilling riser and the assessment of understanding the VIV performance differences obtained when using staggered bare and buoyant joints along the riser. It is a generally accepted view that staggering bare joints improves riser VIV response however current analysis techniques do not provide the capability to evaluate the staggering and no hard evidence exists (e.g. test data) in support of a staggering philosophy. Through the use of CFD the influence of the non-cylindrical bare joints is captured in the analysis process permitting at least a qualitative evaluation of particular staggered configurations. This application and the stated objectives of the work are used to highlight the main points of the preceding discussion.

Introduction

The last years have seen an increasing interest in the use of Computational Fluid Dynamics for the assessment of the Vortex-Induced-Vibration (VIV) behaviour of marine risers. This is still an on-going process where CFD methods are undergoing rigorous testing to prove their applicability in marine riser design. CFD is widely used in other industries (automotive, aerospace, energy) but investment has been made to achieve sufficiently good accuracy in the models for the various industryspecific flow regimes encountered. The offshore/subsea industry is now starting to do the same in realising the potential for CFD application to VIV, process simulation, splash zone loading etc. Currently, project application of CFD is mainly in complimentary role. Deepsea uses CFD for several types of projects such as workover riser VIV, strake and fairing design/analysis, as well as riser tower VIV analysis and drilling riser configuration development.

This paper aims to outline the procedure used for the development of the configuration of a drilling riser using CFD analysis for the assessment of the hydrodynamic behaviour of the drilling riser components which is being used as input to standard industry-accepted tools for frequency domain VIV analysis (Shear7).

Key CFD Modelling Aspects

There are several levels of complexity, resource expense and accuracy with respect to CFD modelling. This paper focuses on the use of the RANS (Reynolds Averaged Navier Stokes) approach which is a good combination of efficiency and accuracy required for use in commercial work. The DES approach combines features of RANS formulations and LES methods. It was developed in an attempt to improve the predictive capabilities of turbulence models in highly separated flows and it is based on the idea of covering the boundary layer using a RANS approach and switching to a LES mode in detached regions. The DES method is at least an order of magnitude more computationally intensive than RANS methods and two-dimensional or axisymmetric calculations are not feasible.

LES (Large Eddy Simulation) solves the instantaneous Navier Stokes equations full instantaneous Navier-Stokes equations up to a prescribed “large scale” and uses “subgrid scale” models for the small-scale motion using appropriate filters to remove very fine time and length scales. LES

methods can give details on the structure of turbulent flows, such as pressure fluctuations which cannot be obtained by RANS methods.

Turbulence Models

This section addresses arguably one of the most critical aspects of any CFD model – the turbulence model. When describing fluid flow mathematically, the flow velocity may be decomposed into two components; mean (U) and fluctuating component (u'). Applying the velocity decomposition to the time-averaged governing equations of the fluid flow (Reynolds averaging) results in the presence of the mean value of the cross products of the fluctuating components of the velocity ($\overline{u'v'}$), typically referred to as Reynolds stresses in the equation which describes the conservation of momentum of the fluid. The effect of this part of the equation can be very important especially as Reynolds number increases. The vast majority of engineering flows are turbulent and as such their simulation needs to include a model to take into account of the effect of turbulent stresses in the distribution of momentum in the flowfield.

Below are presented some of the most popular turbulence models which are of the “eddy viscosity” family. In this family of models, the Reynolds stresses are related to the mean gradients of the velocity in the flow field, thus represented as increased viscosity via the Boussinesq approximation. The turbulence models seek to “close” the set of flow equations by calculating the eddy viscosity. For clarity, it has to be said that in contrast to dynamic or kinematic viscosity, eddy viscosity is a property of the flow and not of the fluid.

Spalart-Almaras

This model calculates the eddy viscosity by solving one additional scalar transport equation for a quantity which is identical to the turbulent kinematic viscosity except in the near-wall (viscous-affected) region. The model accounts for the production and destruction of turbulent energy (in the near-wall region) and has been used successfully in aerodynamic flows extensively.

k- ϵ Models

The family of k- ϵ models is probably the most popular turbulence models to date. However, due to the increase in computational efficiency and some of their inherent deficiencies has led to the switch of interest towards more sophisticated turbulence models or approaches (such as LES and DES) in the past 5 years. In k- ϵ models the calculation of the “turbulent viscosity” is made through the solution of two equations in addition to the mean fluid flow equations. These equations represent the conservation of turbulent kinetic energy (k) and turbulent energy dissipation (ϵ) in the flowfield, thus referred to as “2-equation” models. Exhaustive descriptions of the specifics of the k- ϵ models can be found in CFD textbooks and fall beyond the scope of this paper which aims to present the relative performance and applicability of each model for VIV applications.

For VIV analyses, assigning freestream boundary conditions is relatively easy with k- ϵ models and also has superior performance in free-shear flows. However, k- ϵ models fail to predict anisotropy in the Reynolds stresses and consequently complex flow (re-circulating, impinging, 3-D). Effectively in k- ϵ models, the whole stress tensor is described by a single scalar (the eddy viscosity).

k- ω Models

k- ω models also belong to the family of two-equation “eddy viscosity” models, where in addition to the mean flow equations, two additional equations for the transport of turbulent kinetic energy (k) and turbulent frequency (ω) are solved.

The main problem of k- ω models has been their sensitivity to freestream boundary (inlet) conditions (ω cannot be defined physically), but its main advantage is that $\omega = 0$ at the wall, which makes things easier on the modelling side and significantly outperforms k- ϵ models on boundary layer flows.

Shear Stress Transport (SST) model The SST model is a hybrid of the k- ϵ and k- ω models. The development of the SST model targeted these deficiencies of the k- ϵ and k- ω models (and the previous attempts for their combination- BSL k- ω model) and features a “limiter” for the equation of turbulent viscosity which takes into account of the transport of turbulent shear stress which has been the weak point of the eddy viscosity models (over-prediction of eddy viscosity at the onset of separation). This feature makes the SST very accurate for the prediction of the onset and amount of flow separation under adverse pressure gradient conditions. It has been shown in the literature through studies of flows around Ahmed bodies, 3D ship hulls etc. that it provides far superior results than 1-equation models (e.g. Spalart- Almaras) as well as k- ω or k- ϵ based models.

This model seeks to close the Reynolds averaged Navier-Stokes (RANS) equations by combining the favourable characteristics of both the k- ϵ and k- ω two-equation models. Essentially the SST uses the k- ω model in near wall regions and k- ϵ in areas that are in fluid dynamic terms far from walls. The logic to this approach is based on the fact that k- ϵ has a number of well known deficiencies in near wall regions which k- ω does not suffer from, whereas this situation is reversed in locations further away from wall boundaries.

Others

There are a host of other turbulence models that have been developed by academia and companies, some showing significant promise in the accuracy and fast calculation, but these require considerable use/testing in their field of application to obtain the general acceptance that they satisfactorily predict the requisite behaviour.

Grid Density & Timestep

It is essential that at the beginning of any CFD work all reported results are demonstrated to be grid and timestep independent. Grid independence is generally first based on mesh density from previous experience that provides grid independent results for bare pipe flows. The mesh density is gradually increased in the region close to the structure to find the mesh density that ensures grid independence.

Timestep independence is based on a characteristic time scale of the problem.

- In forced oscillation problems, usually the timestep providing timestep independent results is some small fraction of the oscillation period or the minimum theoretical Strouhal (shedding) period of the elements of the structure (whichever is smaller).
- In free oscillation and static problems, the timestep is some small fraction of the minimum theoretical Strouhal period of all elements of the structure.

Boundary Conditions

Assigning boundary conditions properly is a critical component of the CFD modeling process. In two-dimensional applications (as the one presented in this study) the computational domain is enclosed between six surrounding surfaces (far-field boundaries) and the wetted body surface (wall boundary). The two surfaces parallel and in-plane with the flow are symmetry boundaries to achieve two-dimensionality, inlet and outlet boundaries are assigned at the furthest upstream and downstream boundaries respectively, while far-field boundaries (prescribing pressure or symmetry conditions) are assigned to the two boundaries parallel and perpendicular to the flow.

Typically the in-flow boundary (inlet) needs to be located at a sufficient distance upstream of the wetted body, such that the flow does not exit from the inlet boundary after impinging on the wetted body, while the outlet boundary (typically constant pressure prescribed) needs to be located sufficiently upstream such that it does not affect the development of the flow, as it serves effectively as a far-field boundary. The locations of the inlet and outlet boundaries are typically determined from boundary location independence studies. However, a plethora of papers have been published on flows around bodies and typical distances for the positioning of the far-field boundaries can be easily found. Certainly these positions are case dependent, so a study is always advisable when the geometry of the body departs significantly from the circular, elliptical or aerofoil-like shape.

Of particular importance for the calculation of hydrodynamic forces on bodies is the wall boundary that describes the contour of the body in the computational domain. Wall boundaries pose a difficult challenge to turbulence models and therefore their treatment is quite complex in comparison to other boundaries. In RANS-based turbulence modelling there

are two main approaches to modelling the boundary layer flow,

- Low-Reynolds number models
- Wall functions

The low-Reynolds number approach resolves the high velocity gradients in the boundary layer and therefore requires a large amount of grid in this area, typically at least twenty nodes across the boundary layer thickness. This approach provides accurate results for both laminar and turbulent boundary layers however there is a high computational overhead associated with it.

Wall functions, on the other hand, use empirical correlations for turbulent boundary layers without resolving the flow pattern near to the wall thus saving computational effort.

Roughness

The roughness of the riser external surface is typically modelled by using a wall function approach. The change of the velocity profile in the near-wall region is modelled according to a correlation which utilises the Equivalent Sand Grain Roughness (ESGR) of the pipe. The ESGR is defined as the height of the closely-packed sand-grains that if they were covering the surface, they would provide the same flow resistance as the rough surface itself. ESGR is effectively the flow-equivalent roughness of the technical (physical) roughness. As such, an ESGR can be assigned to any technical roughness.

Ideally, in order to determine the ESGR of a rough surface, an experiment (real or numerical) should to be carried out to determine the velocity distribution on the rough wall. From this, the closely-packed sand-grain height that would provide the same velocity profile can be found. The diameter of that sand-grain would be the ESGR. ESGR tables for technically important wall surfaces can be found in the literature (DIN 1952) otherwise it is necessary to calculate ESGR either by extrapolating from experimental data available in the literature, performing numerical/physical experiments or by calculating it based on roughness elements volumes.

As a wall function approach is used, the whole of the boundary layer is assumed to be turbulent, but given that roughness has the effect of causing an earlier transition to a turbulent boundary layer this is a reasonable assumption.

Validation & Benchmarking

There is a strong need for validation of CFD simulations to increase the confidence in the results produced by such calculation methods. Generally for non-standard riser configurations, such as workover risers and riser towers with external risers, model testing is conducted to generate the lift coefficient map for use in standard VIV tools such as Shear7.

With the collection of this data comes the opportunity for model validation and determination of accuracy of the CFD predictions.

Benchmarking is an extremely useful way of providing enhanced QA and confidence in the CFD predictions. Benchmarking is required for CFD applications so that the accuracy of particular codes/turbulence models can be assessed against a known, industry-accepted problem and results data set.

Defining acceptable levels of accuracy is related to the purpose of the CFD work. If it is for qualitative evaluation of trends, design modifications etc., which cannot feasibly be assessed with model testing (economic and schedule constraints), accuracy of $\pm 50\%$ is acceptable provided the trend predictions, location of peak lift values and the crossing points from excitation to damping are close. Should CFD be used for quantitative purposes then the accuracy of magnitude, sign and the trends are more important. Deepsea has found CFD generally under-predicts the magnitude of lift coefficients between -5% and -30% , depending on whether at peak response, so sensitivity analyses are required to establish the impact of the potential inaccuracy of the CFD results on the findings of any particular study.

Case Study

An area well suited for CFD application is that of a deepwater drilling riser – through the use of CFD the influence of the non-cylindrical bare joints is captured in the analysis process permitting at least a qualitative evaluation of particular staggered configurations. The purpose of staggering bare and buoyant joints in the upper part of the riser is such that the bare joints disrupt the consistent vortex shedding pattern set up around a continuous length of buoyant, cylindrical joints. In fact most bare joints are not purely cylindrical in profile and have choke and kill lines, control umbilical(s) and other auxiliary lines around the main riser pipe which give the non-cylindrical profile. These profiles are also typically asymmetric, therefore the behaviour of the cross-section with respect to the exciting hydrodynamic forces may be heavily direction-dependent.

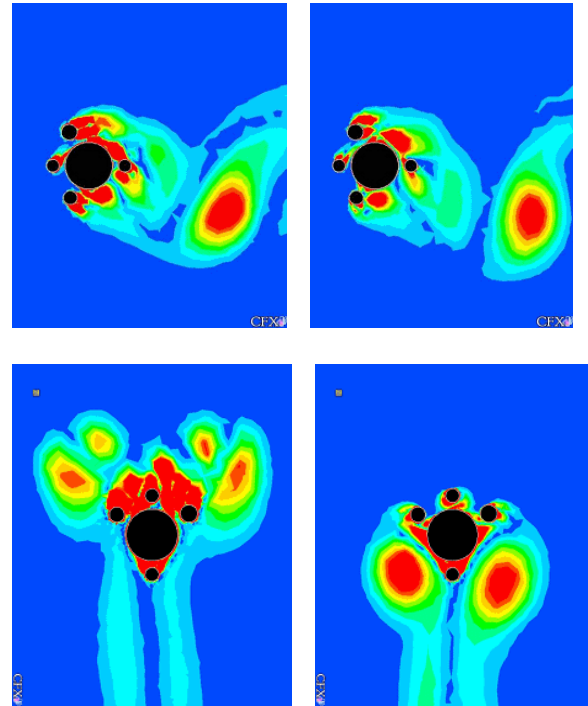
The cross-section profile of the bare joints tends to disrupt the regular vortex shedding flow around the section so rather than providing lift in a particular flow regime, as circular buoyant joints would, the slick joints sometimes produce no lift or, often, damping. This different behaviour is why staggering joints is an effective method of gaining some improvement in VIV performance with little or no cost (only changes the order in which joints are run with no capital equipment costs).

VIV analysis is traditionally made using two methods of analysis:

- Frequency domain analysis
- Time domain analysis

The former entails the calculation of the natural frequencies of mode shapes of the submerged structure and calculation of the excitation forces and their contribution to the excitation of the various modes of the structure. The standard industry tool for such calculations is Shear7, a VIV analysis tool package based on frequency domain analysis developed by MIT. In this approach, the accuracy of the model depends fully on the level of accuracy on the representation of the non-circular cross sections of the structure. To this end, two-dimensional CFD models are constructed for each cross-section and the “worst” (least damping or most exciting) orientation is investigated in forced oscillation in typical flow conditions. Due to the asymmetry of the geometry, the orientation of the flow clearly alters the flowfield pattern and the characteristics of the hydrodynamic forces exerted on the structure (Figure 1).

Upon completion of this study, further forced oscillation runs are conducted in the “worst orientation” of the flow to map the hydrodynamic behaviour of the cross-section in various oscillation amplitudes, frequencies and flow conditions. These quantities are usually grouped in non-dimensional quantities such as reduced velocity, V_r , which is the ratio between the flow velocity and the product of the forced oscillation frequency and hydrodynamic diameter and amplitude-to-diameter ratio.



**Figure 1: Flow patterns for forced oscillations in different incidence angles;
(a) 0 degrees (above), (b) 90 degrees (below)**

The complete description of the hydrodynamic behaviour entails the maps of the lift coefficient in-phase with oscillation velocity in the V_r - A/D space. These maps can be significantly different from the ones produced for circular cylinders and their representation is important for the accuracy of the model. Perhaps the most important disadvantage of the Shear7 approach is that it includes only transverse motion of the riser and has no capability to include the effect of the in-line motion which can have a significant influence on the behaviour of the riser.

A more detailed approach with inclusion of the in-line motion is provided by full time domain analysis. This entails the solution of the coupled flow-structural equations for the whole riser in the time domain. From that, FFT analysis on the results can yield the riser response as well as the most excited modes of the structure. However, the computational requirements for such a task are immense which renders a full 3-D calculation of a riser impossible. As an alternative, the strip-method approach has proved very effective. In the strip-method approach, the flow is solved on a number of pre-selected two-dimensional slices (strips) along the riser length and is two-way coupled with an FE model which calculates

the riser dynamics in a semi-implicit manner. This type of solution, although computationally significantly more demanding, can yield more information on the interaction of the fluid and the riser as sections of the flowfield are resolved.

In the case study presented here, frequency domain analysis has been used for the calculation of two risers. The two risers have the same length of bare (slick, asymmetric) and buoyant joints (of circular cross-section) but are arranged in a different manner. Riser 1 features the whole of the 1200 ft of slick joints at the bottom of the riser, while in Riser 2 the joints are equidistantly redistributed along the riser length. CFD analysis has been used to generate the Vr-A/D maps of the lift coefficient of the slick cross-sections. The profile of the in-coming current is non-uniform.

	Riser 1	Riser 2
Description	14 x 75 ft slick jts 50 x 75 ft buoyant jts	14 x 75 ft slick jts 50 x 75 ft buoyant jts
Joint Arrangement	All slick joints located at the bottom	Staggered along length of riser (3x buoyant jts, 1x slick jt)

Table 1: Case Study – Joint arrangements for Risers 1 and 2

The generation of the A/D-Vr maps of the lift coefficients of the slick joints showed damping behaviour for all forced oscillation conditions (Figure 2). In order to use a more conservative approach, the maps have been represented by a line parallel to the A/D axis, hence the lift coefficient is zero for all oscillation amplitudes.

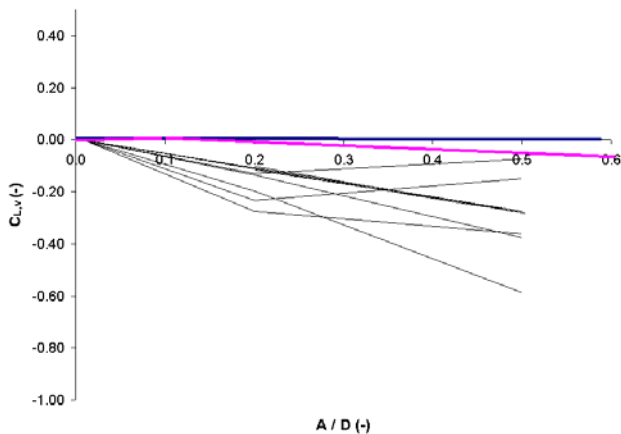


Figure 2: Lift curves used for slick joints in Shear7

The results from the Shear7 analysis are presented in terms of RMS of displacement and RMS of stress in Figure 3. The results clearly show the benefits of using staggered slick joints along the length of the riser via the reduction the RMS stresses and therefore the increase in fatigue life of the structure as a whole. The model succeeds in capturing the beneficial damping effect of the slick joints and more importantly is sensitive enough to highlight the differences in the riser response due to the non-uniform mass and damping distribution. This procedure therefore is particularly useful for parametric studies for the optimization of the riser without additional cost, merely using re-distribution of the slick joints. The optimal distribution of the slick joints certainly depends

on metocean data and should be performed independently for each drilling site.

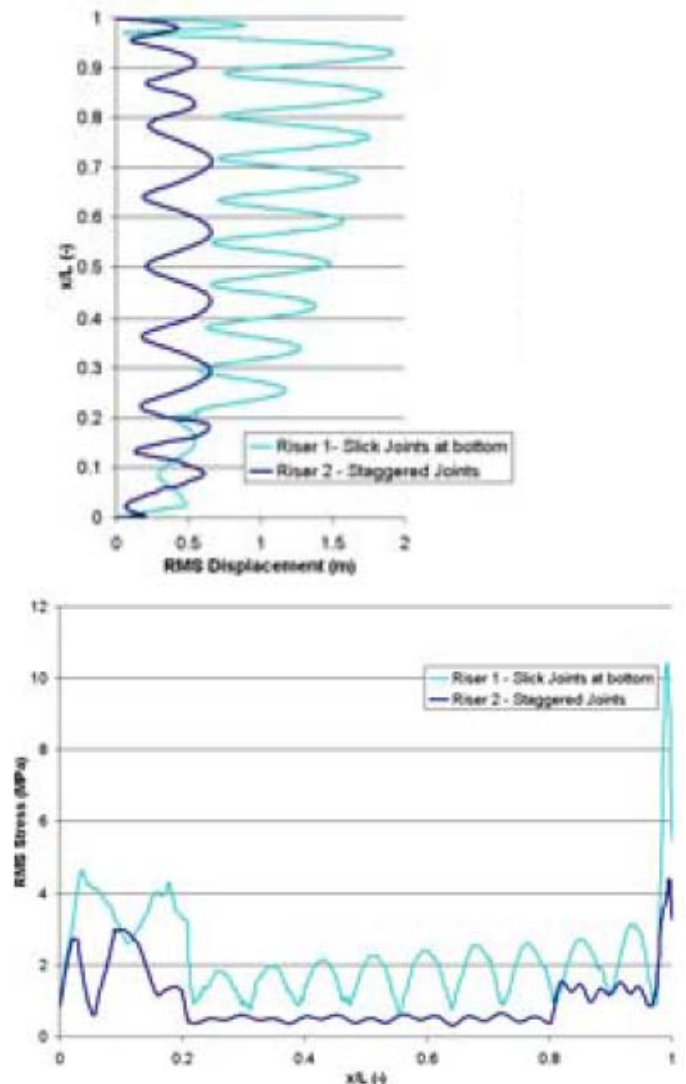


Figure 3: RMS Displacement and RMS Stress along the riser length for Risers 1 and 2

Conclusions

CFD is a numerical process that is beginning to be more widely considered for project applications in the offshore arena. To accelerate the acceptance and usage of such tools it is necessary that consistent benchmarking practises are established to provide an industryaccepted framework in which CFD application can develop and mature.

Deepsea has worked on a number of deepwater projects in which CFD analysis has formed the main modelling approach. An example of this is the determination of whether staggering slick joints in a drilling riser configuration is an effective method of gaining some improvement in VIV performance with little or no cost (only changes the order in which joints are run with no capital equipment costs). Given the expense and necessary timeframe to perform comprehensive model testing CFD combined with frequency analysis provides a viable alternative to assist in the design and analysis of the VIV response of deepwater marine risers.

References

- [1] Launder, B. E. and Spalding, D.B. The numerical computation of turbulent flows. *Computer Methods in Applied Mechanics and Engineering*, 3: 269-289, 1974.
- [2] Launder, B.E., Reece, G. J. and Rodi, W. Progress in the development of a Reynolds stress, turbulence closure. *J Fluid Mech*, 68: 537-566, 1975
- [3] Menter, F.R. Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*, 32(8), 1984
- [4] Menter, F.R. Multiscale model for turbulent flows. *AIAA 24th Fluid Dynamics Conference, AIAA*, 1986.
- [5] Menter, F. R. and Kuntz, M. Development and application of a zonal DES model for CFX-5. *CFX validation report*, CFX-VAL 17/0503.
- [6] Menter, F. R. and Kuntz, M. Adaptation of eddy-viscosity turbulence models to unsteady separated flow behind vehicles. *Proc. Conf. The aerodynamics of heavy vehicles: trucks, buses and trains*, Asilomar, Ca, 2002.
- [7] Shlichting, H. Boundary Layer Theory. *McGraw-Hill*, 1979
- [8] Wilcox, D.C. Turbulence Modelling for CFD. *DCW Industrie*, 1986.
- [9] Wilcox, D.C. Multiscale model for turbulent flows. *AIAA 24th Sciences Meeting, AIAA*, 1986.