VI International Conference on Computational Methods for Coupled Problems in Science and Engineering
COUPLED PROBLEMS 2015
B. Schrefler, E. Oñate and M. Papadrakakis(Eds)

A NEW VERY LARGE EDDY SIMULATION MODEL IN THE CONTEXT OF FLUID-STRUCTURE INTERACTION

A. KONDRATYUK*, **M. SCHÄFER**†

* Graduate School of Computational Engineering (GSC CE)
Technical University Darmstadt
Dolivostrae 15, 64293 Darmstadt, Germany
e-mail: kondratyuk@gsc.tu-darmstadt.de, web page: http://www.graduate-school-ce.de

† Institute of Numerical Methods in Mechanical Engineering
Technical University Darmstadt
Dolivostrae 15, 64293 Darmstadt, Germany
e-mail: schaefer@fnb.tu-darmstadt.de, web page: http://www.fnb.tu-darmstadt.de

Key words: Turbulent Flow, Very Large Eddy Simulation, Hybrid Methods, Fluid-Structure Interaction

Abstract. In the present work one of the hybrid VLES (Very Large Eddy Simulation) turbulence models is investigated in the context of fluid-structure interaction (FSI). Firstly, the formulation of the VLES model for two different RANS models ($k-\varepsilon$ and $\zeta - f$) is validated with a fully-developed channel flow at a turbulent Reynolds number of $Re = 395$. Then, this model is used to calculate the flow over an inclining plate in order to investigate the potential of VLES for moving structures. The results of simulations using two different background RANS models are compared to URANS and DDES results. In addition, the simulation results for different underlying RANS models are discussed.

1 INTRODUCTION

Direct Numerical Simulation (DNS) of turbulence can only be applied to flows with very simple geometries and small Reynolds numbers, because a complete resolution of the turbulent structures is necessary. Another possibility is Large Eddy Simulation (LES), where the big energy containing turbulent scales are resolved, while the small ones are modeled. In this case the computational costs increase very quickly with increasing Reynolds number. This is why in many industrial fields Reynolds-Average-Navier-Stokes (RANS) models are still the preferred method for the prediction of turbulent flows. In this approach all turbulent structures are modeled. Therefore, the results obtained with RANS models are often not satisfactory for many kind of flows, in particular, for massively separating flows.

In the last decade so-called hybrid turbulence models became increasingly popular. Compared to LES and DNS they deliver satisfactory results while demanding reduced computational costs. The underlying idea is to combine the advantages of different modeling approaches. These new models make it possible to solve also complex industrial problems.

The most popular hybrid turbulence model which has also been successfully used for many complex turbulent flow tasks is the Detached Eddy Simulation (DES), which was first proposed by Spalart [11]. It combines a RANS mode in the attached boundary layers with LES in separated regions and regions far from the wall. The complication in the applying of DES is the "gray area", in which an undefined modeling zone exists. In this area the solution is neither pure RANS nor pure LES [5].

Another kind of hybrid methodology, the so-called Very Large Eddy Simulation (VLES), was proposed by Speziale [12]. This model provides a seamless change from RANS to DNS depending on the numerical resolution. However, the original VLES model damped the Reynolds stress too much and required a fine mesh resolution. Therefore, modification were proposed in [5] or [3], with which the approach shows high efficiency and robustness in many applications [6], [5], [3].

In [4] it has been reported, that the predictive accuracy of VLES depends on the specific RANS turbulence model. This aspect is particularly observable for complex flows with movable or deformable objects. This especially occurs for flows with separations and thus justifies the use of more complex RANS models, like the $\zeta - f$ model.

The turbulent fluid-structure interaction (FSI) is currently not an established research object. However, especially in the case of FSI the problems become much more demanding in terms of computational cost owing to additional equations of motion for the structural part, which have to be solved together with the flow equations within a coupled solution procedure. Therefore, a reduction of computing times is especially important in the context of FSI. The potential of hybrid modeling employed to FSI is mentioned in [14], while some experiences are described in [2]. In these works the DES model has been used and investigated. Studies on the behavior of the VLES model in the context of FSI are rare.

In this paper $k - \varepsilon$ and $\zeta - f$ VLES models are investigated in the context of FSI. First, the VLES models are validated by computing the attached flow in a channel at a Reynolds number of $Re = 395$. Finally, the models are applied to investigate a flat plate which inclines at a constant angular velocity from $0°$ to $45°$ at $Re = 30000$. The focus lies on the investigation of the results from the VLES model with different basic RANS models and the comparison of the results to DES and URANS results.

2 GOVERNING EQUATIONS

2.1 Reynolds Averaged Navier Stokes equations

In the present study an incompressible Newtonian fluid with constant fluid properties is considered. The Navier-Stokes equations describe the conservation of mass and momentum for such fluids. Using the Reynolds decomposition and time averaging these equations lead to the so-called RANS equations with an additional term, which arises due to the averaging:

$$\frac{\partial \overline{u_i}}{\partial x_i} = 0 \,, \tag{1}$$

$$\frac{\partial \overline{u_i}}{\partial t} + \overline{u_j}\frac{\partial \overline{u_i}}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\nu \frac{\partial \overline{u_i}}{\partial x_j} - \tau_{ij}\right) - \frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_j}, \tag{2}$$

where $\overline{u_i}, \overline{p}$ and ν are mean velocity components, mean pressure and viscosity, respectively. τ_{ij} represents the Reynolds stress tensor, which can be represented by the Boussinesq's approximation as

$$\tau_{ij} = -\overline{u_i' u_j'} = 2\nu_t S_{ij} - \frac{2}{3}k\delta_{ij}, \quad S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right). \tag{3}$$

Here ν_t presents a turbulent viscosity, which has to be modeled. In this paper only the $\zeta - f$ model is presented, a detailed description of the $k - \varepsilon$ model can be found in [8].

2.2 $\zeta - f$ model

The $\zeta - f$ RANS model developed by Hinjalic et al. [7] uses a transport equation for the velocity scales ratio $\zeta = \overline{v^2}/k$ and the equation of the so-called elliptic relaxation function f, additionally to the equations for turbulent kinetic energy k and its dissipations rate ε:

$$\frac{\partial k}{\partial t} + u_j\frac{\partial k}{\partial x_j} = P_k - \varepsilon + \frac{\partial}{\partial x_j}\left[\left(\nu + \frac{\nu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right)\right], \tag{4}$$

$$\frac{\partial \varepsilon}{\partial t} + u_j\frac{\partial \varepsilon}{\partial x_j} = \frac{C_{\varepsilon_1}P - C_{\varepsilon_2}\varepsilon}{T} + \frac{\partial}{\partial x_j}\left[\left(\nu + \frac{\nu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_j}\right)\right], \tag{5}$$

$$\frac{\partial \zeta}{\partial t} + u_j\frac{\partial \zeta}{\partial x_j} = f - \frac{P}{k}\zeta + \frac{\partial}{\partial x_j}\left[\left(\nu + \frac{\nu_t}{\sigma_\zeta}\frac{\partial \zeta}{\partial x_j}\right)\right], \tag{6}$$

$$L^2\Delta f - f = \frac{1}{T}\left(C_1 + C_2\frac{P}{\varepsilon}\left(\zeta - \frac{2}{3}\right)\right). \tag{7}$$

The corresponding turbulence viscosity is defined as

$$\nu_t = C_\mu^\zeta \zeta k T, \tag{8}$$

where T is the turbulent time scale and C_μ^ζ is a model constant. The coefficients and a detailed description of this model can be found in [6].

This eddy-viscosity-based model yields better results in comparison to other RANS models for the wall-bounded flows [6]. Since the predictive accuracy of VLES depends on the specific RANS turbulence model [4], the application of the $\zeta - f$ model as a background RANS model for VLES appears to be promising.

3 VLES MODEL

The VLES approach switches from RANS to DNS depending on the numerical resolution. Between these two limits a LES will be recovered [3]. The switching is realized through the rescaling of the subscale stress resolution control function F_r that is introduced in this approach:

$$\tau_{ij}^{sub} = F_r \tau_{ij}^{RANS}. \tag{9}$$

F_r is a function of two length scales: the turbulent length scale L_c related to the spectral cut-off and the integral length scale L_i ($\propto k^{3/2}/\varepsilon$):

$$F_r = min\left[1, \left(\frac{L_c}{L_i}\right)^{\frac{3}{4}}\right]. \tag{10}$$

F_r gets a value between one and zero. When F_r approaches 0, then all scales are resolved and the VLES model behaves like a DNS. In the near-wall region $F_r \to 1$, because $L_c > L_i$ and the model works as a RANS model, what is similar to the DES concept. In [3] a detailed description of the VLES approach and the resolution control function F_r can be found.

The VLES model can be blended with any trusted RANS turbulence model. In this paper it was implemented with the standard $k-\varepsilon$ model [8] and with the $\zeta-f$ model [6] described above. Compared to basic RANS models the VLES modifies only the formulation of the turbulent viscosity. For example, for the $\zeta - f$ model the turbulent viscosity takes the form

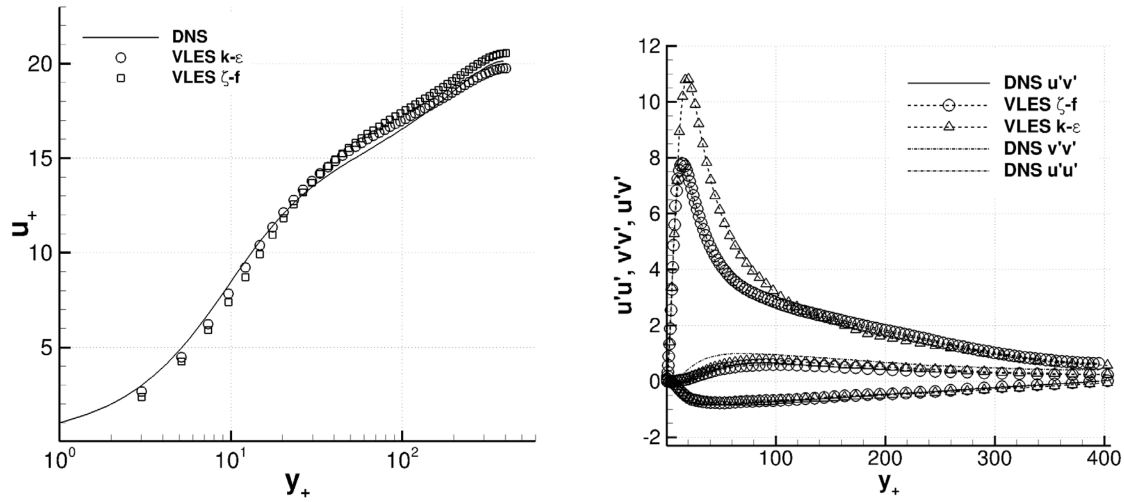
$$\nu_t = F_r C_\mu^\zeta \zeta k T. \tag{11}$$

In [5] and [3] it has been shown that the VLES approach is capable of achieving good predictions for a wide range of turbulent flows with less computational effort in comparison to LES.

4 VALIDATION

The VLES model described before is validated with a fully-developed channel flow at a turbulent Reynolds number of $Re_\tau = 395$, based on the friction velocity u_τ. This test case shows the feasibility of the VLES model to predict the attached boundary layer flows.

The size of the computational domain is given by $L_x = 2\pi, L_y = 2$ and $L_z = \pi$, for streamwise, wall-normal and spanwise direction, respectively. For the simulation a mesh with 64x100x40 grid points is used. The first grid node is located at the normalized wall distance $y^+ = 1$. Periodic boundary conditions are applied in streamwise and spanwise



(a) Mean streamwise velocity profiles (b) Reynolds stress component profiles

Figure 1: Comparison of results for channel flow

directions. The results of the calculation within the VLES model for two different basic RANS models, $k-\varepsilon$ and $\zeta-f$, are compared to the DNS data contributed by Moser [10].

Figure 1a shows the mean streamwise velocity given by different background RANS models. Some difference in the buffer layer and the log-low region can be seen. However, VLES shows good agreement to the reference for both overlying RANS models considering that the mesh is quite coarse. The RMS velocities are compared in Figure **??** (b). The VLES with the basis $\zeta-f$ model predicts the results very well in all of the three directions. The $k-\varepsilon$ shows good agreement in the $v'v'$ and $u'u'$ components, while the values for $u'v'$ are overpredicted in the buffer layer. It can be seen, that the results of the VLES models for the turbulent channel flow are in good agreement with the DNS prediction.

5 NUMERICAL METHODS AND COMPUTATIONAL DETAILS

Next, the flow around a flat plate, which is inclining from 0° to 45°, is investigated with the VLES model. The plate is mounted inside a plane channel. The computational domain is shown schematically in Figure 2. It is the same configuration as in [15]. The length of the computational domain corresponds to 2 m, the height and the depth of the channel are equal to 0.45 m. The chord length and the thickness of the plate is $c = 0.12\,\mathrm{m}$ and 0.006 m, respectively, the aspect ratio is $AR = 3.67$.

The plate is placed in the center of the channel and changes its angle of attack rapidly from 0° to 45° at an averaged rotational speed of

$$\alpha = \frac{10\pi}{3}\frac{\mathrm{rad}}{\mathrm{s}}. \tag{12}$$

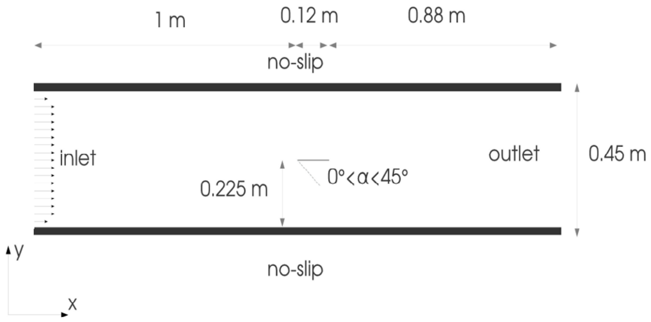


Figure 2: Sketch of computational domain

For the description of this motion the following equation is used:

$$\alpha(t) = \frac{\pi}{4}\left[\sin\left(\frac{\pi}{T}t\right)\right]^2, \tag{13}$$

where $T = 0.15\,\mathrm{s}$ equals one period. This expression makes it possible to avoid the discontinuities in the calculations. The reduced pitching frequency for a non periodic motion is defined as $k_{pitch} = \Delta\alpha c/(U_b \Delta t) = 0.168$, where $U_b = 3.75\,\mathrm{m/s}$ is the inlet bulk velocity.

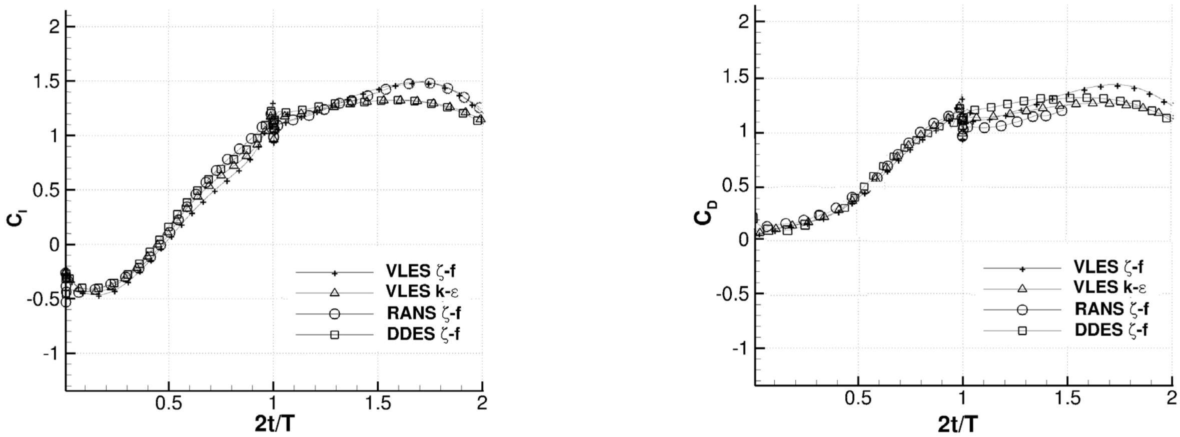
This simulation was carried out at a Reynolds number of $Re_c = 30000$. Non-slip boundary conditions are applied at the surface of the plate as well as at the top and the bottom of the computational domain. A profile of a fully developed turbulent channel flow without any perturbations is prescribed at the inlet. At the outlet, a zero gradient boundary condition is applied. Furthermore, periodic boundary conditions in the spanwise direction (z-axis) are assumed.

For all simulations the same grid with about 2.6 million CVs is applied, with 240 CVs in circumferential direction of the plate. The first node in normal grid direction is located at $y^+ = 1.0$. The mesh is clustered around the plate and this block rotates with the plate. This ensures a good grid quality in the region, where the vortex shedding and recirculation is expected. This allows to lower the computational cost, because no re-meshing and no additional grid generation methods are required in this region.

All simulations are carried out with the code FASTEST [13], which is based on the finite-volume method for block-structured grids. The parallelization in FASTEST is done via domain decomposition using MPI. For the approximation of convective and diffusive fluxes the central scheme of second-order accuracy is applied. A second-order backward differencing scheme is used for the time discretization. The coupling between pressure and velocity is done with the SIMPLE algorithm.

6 RESULTS AND DISCUSSION

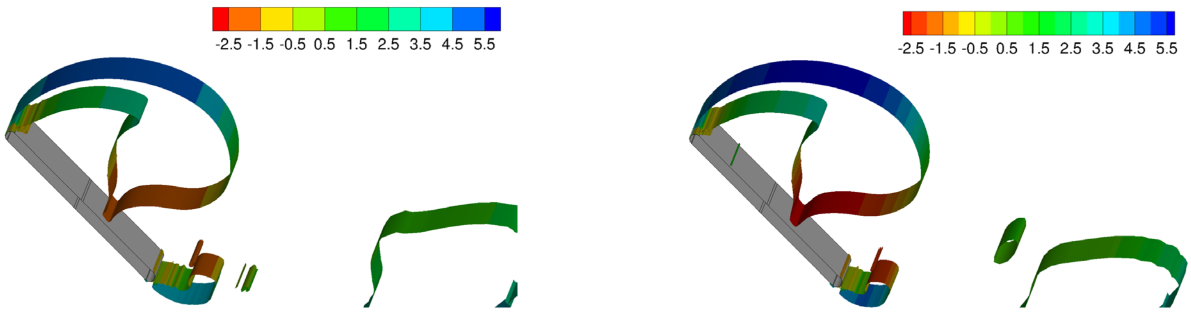
The flow over an inclined flat plate is simulated with the $\zeta - f$ and $k - \varepsilon$ VLES models. The character of the flow and the generation of the fluctuation for the quickly moving structures are investigated.



(a) Lift coefficient for $0 \leq t \leq T/2$ (b) Drag coefficient over $0 \leq t \leq T/2$

Figure 3: Comparison of lift and drag coefficients

In the beginning of the simulation the plate changes its angle of attack from 0° to 45°. In this upstroke phase no outstanding fluctuations are generated and the flow has a two-dimensional character. This effect corresponds to the results by Martian [9], who investigated the flow past a pitching NACA0012 airfoil. In this phase VLES, DDES and URANS predict similar values for the lift C_L and drag C_D coefficient (Figure 3).



(a) VLES $\zeta - f$ for $t = T/2$ (b) VLES $k - \varepsilon$ for $t = T/2$

Figure 4: Fluctuation visualized by Q-criterion at $Q = 5$, colored by streamwise velocity

Afterward, the plate remains at the constant angle of attack $\alpha = 45°$ and the C_L and C_D values deviate (see Figures 3). Between URANS and VLES $\zeta - f$ no significant difference can be detected. In this flow phase fluctuations should start to be generated and become more dominant over time [15]. However, the VLES simulations further shows a two-dimensional character of the flow. Fluctuations have not been developed yet (Figures 4). The lack of fluctuations for hybrid turbulence models is also observed by Türk et al. [15]. There it is also shows, that the URANS approach can predict the three-dimensional character of this flow already in the beginning of the "remain"-phase.

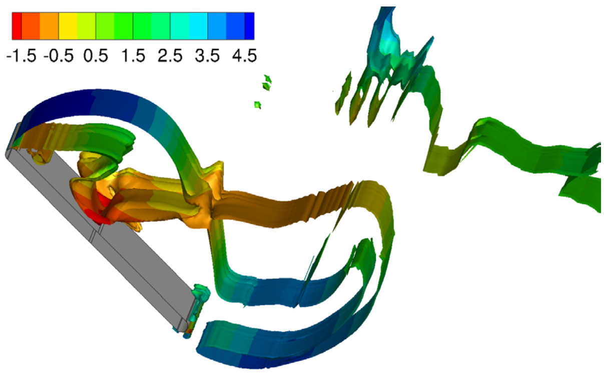


Figure 5: VLES $k - \varepsilon$ for $t = 3T$

Until $t > 3T$, the fluctuations start to form up and the flow shows a three-dimensional character (Figure 5). The values of C_L show a good agreement after this instant. C_L is underpredicted by the VLES $k - \varepsilon$ model in the beginning of the "remain"-phase, but it approaches the values of VLES $\zeta - f$ after a few iterations (Figure 6).

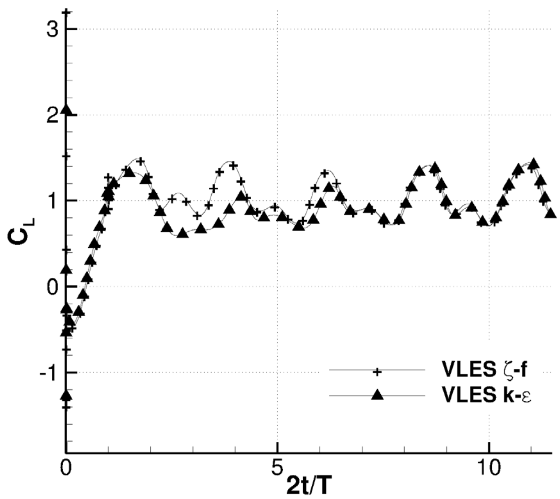
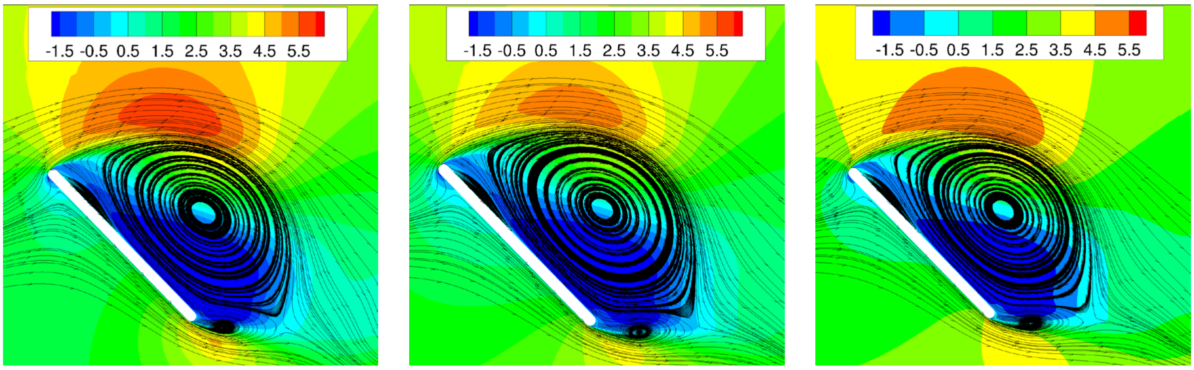


Figure 6: Comparison of lift coefficient for $0 \leq t \leq 6T$



(a) VLES $k - \varepsilon$ (b) VLES $\zeta - f$ (c) URANS $\zeta - f$

Figure 7: Contour and streamlines of streamwise velocity component for $2t/T = 1.8$

The character of the flow past an inclined plate is known from a study by Breuer et. al [1]. A large clockwise rotating vortex exists at the leeward side of the plate. This vortex forms from the flow separation at the leading edge. The small size clockwise recirculation originates at the trailing edge because of a roll-up of the shear layer. VLES simulations predict these overall flow features very well (Figure 7). An insignificant difference appears only in the location of the center of small vortices (Figure 8).

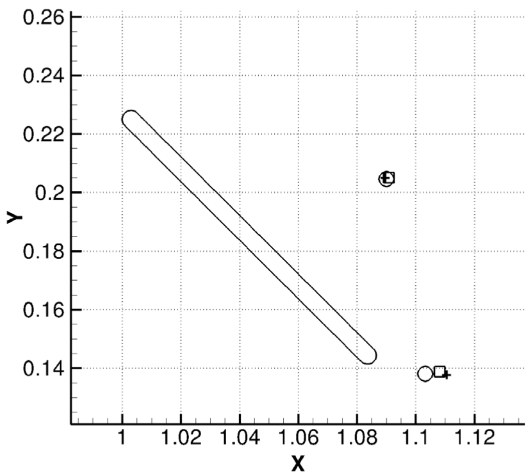


Figure 8: Location of the vortex cores, + VLES $\zeta - f$, $\circ$ VLES $k - \varepsilon$ and $\square$ URANS

7 CONCLUSION

We have presented the new VLES $\zeta - f$ model. We validated this model with a turbulent channel flow at a Reynolds number of $Re = 395$. The validation result shows good agreement with the reference DNS data. Then the flow over an inclined plate was investigated with the VLES approach and compared with URANS results. It was shown that the VLES model yields unsatisfactory results on moving structures. The VLES approach requires some computational time to predict the 3D character of the flow after the plate has come to a rest. However, the character of the flow over the fixed inclined plate was predicted very well by the VLES model.

This study is the first step to investigate the potential of VLES for moving structures. The VLES approach with a modified $\zeta - f$ model can lead to an acceleration in the development of fluctuation. Another factor that needs to be investigated is the influence of the filter width on the results within VLES model.

8 ACKNOWLEDGMENT

This work is supported by the 'Excellence Initiative' of the German Federal and State Governments within the Graduate School of Computational Engineering at Technische Universität Darmstadt.

REFERENCES

[1] Breuer, M., Jovičić, N. and Mazaev, K. Comparison of DES, RANS and LES for the separated flow around a flat plate at high incidence. *International journal for numerical methods in fluids*, 41(4), 357-388, 2003

[2] Bunge, U. Numerische Simulation turbulenter Strömungen im Kontext der Wechselwirkung zwischen Fluid und Struktur. *Univ.-Verlag der TU, Univ.-Bibliothek*, 2005

[3] Chang, C. Y., Jakirlic, S., Dietrich, K., Basara, B. and Tropea, C. Swirling flow in a tube with variably-shaped outlet orifices: An LES and VLES study. *International Journal of Heat and Fluid Flow*, 49 pp. 28-42. 2014

[4] Han, X., and Krajnović, S. "A new very large eddy simulation model for simulation of turbulent flow."*Progress in Hybrid RANS-LES Modeling.* Springer Berlin Heidelberg, pages 131-140, 2012.

[5] Han, X. and Krajnović, S. Validation of a novel very large eddy simulation method for simulation of turbulent separated flow. *International Journal for Numerical Methods in Fluids*, 73(5), pages 436-461, 2013

[6] Han, X. and Krajnović, S. An efficient very large eddy simulation model for simulation of turbulent flow. *International Journal for Numerical Methods in Fluids*, 71(11), pages 1341-1360, 2013

[7] Hanjalić, K., Popovac, M. and Hadžiabdić, M. A robust near-wall elliptic-relaxation eddy-viscosity turbulence model for CFD. *International Journal of Heat and Fluid Flow*, 25(6), pages 1047-1051, 2004

[8] Jones, W. P. and Launder, B. The prediction of laminarization with a two-equation model of turbulence. *International Journal of Heat and Mass Transfer*, 15(2), pages 301-314, 1972

[9] Martinat, G., Braza, M., Hoarau, Y. and Harran, G. Turbulence modeling of the flow past a pitching NACA0012 airfoil at 105 and 106 Reynolds numbers. *Journal of Fluids and Structures*, 24(8), 1294-1303, 2008

[10] Moser, R. D., Kim, J. and Mansour, N. N. Direct numerical simulation of turbulent channel flow up to Re= 590. *Phys. Fluids*, 11(4), 943-945, 1999.

[11] Spalart, P. R., Jou, W. H., Strelets, M., and Allmaras, S. R. Comments on the feasibility of LES for wings, and on a hybrid RANS/LES approach. *Advances in DNS/LES, 1*, pages 4-8, 1997

[12] Speziale, C. Turbulence modeling for time-dependent RANS and VLES: a review.*AIAA Journal*, 36(2), pages 173-184, 1998

[13] Sternel, D. C. FASTEST-Manual, 2005.

[14] Squires, K. D. Detached-eddy simulation: current status and perspectives. *In Direct and large-eddy simulation V*, pages 465-480. Springer Netherlands, 2004.

[15] Türk, S., Reimann, T., Sternel, D. C., and Schäfer, M. On the performance of detached-eddy simulation in fluid-structure interaction problems. *V International Conference on Computational Methods for Coupled Problems in Science and Engineering, Coupled Problems 2013*, , Barcelona, 2013. Proceedings.

[16] Wilcox, D. C. *Turbulence modeling for CFD* (Vol. 2, pp. 103-217). La Canada, CA: DCW industries, 1998.