Proceedings of the ASME 2012 31st International Conference on Ocean, Offshore and Arctic Engineering
OMAE2012
July 1-6, 2012, Rio de Janeiro, Brazil

OMAE2012-84215

WORKSHOP ON VERIFICATION AND VALIDATION OF CFD FOR OFFSHORE FLOWS

L. Eça
Department of Mechanical Engineering
IST
Technical University of Lisbon
Lisbon, Portugal
eca@marine.ist.utl.pt

G. Vaz
CFD Projects
R&D Department
MARIN
Wageningen, the Netherlands
g.vaz@marin.nl

ABSTRACT

This document introduces the Workshop on Verification and Validation (V&V) of CFD for Offshore Flows, to be held during OMAE2012. It presents a brief introduction to the purpose of Verification and Validation with the identification of the goals of code and solution verification and validation. Within this context, three test-cases are proposed: Case-I of code verification, Case-II of solution verification and Case-III of solution verification and validation. Case-I consists on a 3D manufactured solution of an unsteady turbulent flow. Case-II is an exercise on the canonical problem of the infinite smooth circular cylinder flow at different Reynolds numbers. Case-III is a more complex flow around a straked-riser. The participants are asked to perform at least one of these test-cases. The objectives for the three proposed test-cases are presented, together with a detailed description of the numerical settings to be used, and the results to be obtained and sent to the Workshop organization. At the end some considerations on general conditions, paper submission, deadlines, and encouragements are stated.

NOMENCLATURE

$\mathbf{V} = (V_x, V_y, V_z)$	Velocity field $[m/s]$
V_{ref}	Reference velocity $[m/s]$
p	Pressure $[N/m^2]$
p_{ref}	Reference pressure $[N/m^2]$
ρ	Water density $[kg/m^3]$
μ	Water dynamic viscosity $[N.s/m^2]$
ν	Water kinematic viscosity $[m^2/s]$
$\mathbf{F} = (F_x, F_y, F_z)$	Forces $[N]$
L_{ref}	Reference length $[m]$
D	Cylinder/riser diameter $[m]$
L	Cylinder/riser length $[m]$
$A = D \times L$	Projected area $[m^2]$
$Re = \frac{V_{ref} L_{ref}}{\nu}$	Reynolds number $[-]$
$C_D = \frac{F_x}{\frac{1}{2}\rho V_{ref}^2 A}$	Drag coefficient $[-]$
$C_{D_{avg}}$	Time-averaged drag coefficient $[-]$
$C_L = \frac{F_y}{\frac{1}{2}\rho V_{ref}^2 A}$	Lift coefficient $[-]$
$C_{L,D_{max}}$	Maximum of lift and drag coefficient $[-]$
$\Delta C_{L,D}$	Amplitude of lift and drag coefficient $[-]$
$C_{L,D_{RMS}}$	RMS of lift and drag coefficient $[-]$
$C_p = \frac{p - p_{ref}}{\frac{1}{2}\rho V_{ref}^2}$	Pressure coefficient $[-]$
t	Time [s]
T_{ref}	Reference time $[s]$
$St = \frac{L_{ref}}{V_{ref} T_{ref}}$	Strouhal number $[-]$
θ	Cylinder circumferential coordinate $[^\circ]$

INTRODUCTION

Nowadays Computational Fluid Dynamics (CFD) is an engineering tool used in many practical applications, also in the offshore field. But given the results of an application it is not immediately clear what the quality of the solution is. This leads to the need of assessing the credibility of CFD simulations by *Verification and Validation* (V&V), [1–3].

One of the main features of V&V is the distinction between numerical errors *(Verification)* and modelling errors *(Validation)* [1, 2]. This is a fundamental issue to guarantee that a good solution is obtained by the right reasons and not by a fortuitous cancelling of errors. Verification is actually composed of two different activities [1, 2]:

1. *Code Verification* aims at demonstrating that a given mathematical model is correctly implemented, by showing that the numerical error tends to zero (with the correct order of accuracy) with grid and time step refinement.
2. The goal of *Solution Verification* is to estimate an error bar of a given numerical prediction that contains the (unknown) exact solution with a given (usually 95%) confidence.

Both activities are exclusively mathematical, but one requires error evaluation (Code Verification), whereas the other involves error estimation (Solution Verification). Naturally, Code Verification should come before Solution Verification. However, Code Verification has an end (unless the code is modified) whereas Solution Verification is required for all applications of a given code.

On the other hand, *Validation* is a science/engineering activity that intends to quantify the modelling error by comparison to the "real world" (experiments). A proper validation requires the knowledge of the experimental and numerical uncertainties [4]. Therefore, Validation must be preceded by Solution Verification.

Many fields of engineering have realized the importance of V&V [4–8] and several Workshops have been organized in different fields to assess the quality of CFD predictions, as for example the AIAA CFD Drag Prediction Workshops [9, 10], and the three Workshops on CFD Uncertainty Analysis held in Lisbon in 2004, 2006 and 2008 [11–13].

A common feature of all these V&V events is that the consistency between the results of different groups using supposedly equivalent tools is not as good as would be desirable [14]. This led to the introduction of a Code Verification exercise in the Lisbon Workshops [15], which proved to be extremely useful for the assessment of the numerical properties of the RANS solvers that participated [15, 16].

The same remark could be made from a literature review of standard offshore applications, as for example the calculation of the flow around a smooth cylinder [17]. Therefore, a Workshop on V&V for offshore applications is proposed for the forthcoming OMAE 2012 Conference. Three test-cases covering Code Verification, Solution Verification and Validation are proposed with the common feature of being unsteady, incompressible, periodic flows. Although more sophisticated models may be applied in 2 of the proposed cases, for this first V&V initiative in offshore flows the main focus will be on URANS solvers. The participants are asked to perform at least one test-case.

WORKSHOP SETUP

Code Verification

One of the challenges of Code Verification in URANS solvers is the inexistence of analytical solutions. Although this may suggest that it is impossible to perform Code Verification for URANS solvers, the *Method of Manufactured Solutions* (MMS) provides an excellent framework to do it [18]. The idea is simple:

- Define a computational domain.
- Define all the dependent variables of your model analytically, i.e. mean velocities and pressure and all turbulence quantities included in the selected turbulence model.
- Substitute the manufactured flow quantities in the partial differential equations and determine the residual/imbalance of the partial differential equations.
- Add the outcome of the previous step to the differential equations as a source term to remove the imbalance of the arbitrary choice of the manufactured flow field.
- Solve the flow problem with your code and compare the results with the analytical solutions (manufactured).

Such procedure may be applied to any mathematical model used in the calculation of turbulent flows. However, the complexity of the procedure is naturally dependent on the formulation chosen and in some cases it may be more difficult than it seems. For example, in eddy-viscosity models there are some rules that must be obeyed to avoid awkward behaviour of the turbulence quantities [19]. Most RANS solvers take advantage of the fact that production is positive and dissipation is negative in their discretization/linearization procedure. If the MS misses such property it is likely that the flow solver will not converge without modifications. However, from the practical point of view, the changes required by this awkward behaviour are useless.

In the proposed Manufactured Solution (MS) the computational domain is a simple rectangular prism with a no-slip condition applied at the bottom boundary and symmetry conditions at the lateral boundaries (**CASE-I**). The flow mimics a pulsating (periodic) flow separation region on top of a near-wall flow that includes a laminar sub-layer and a wall shear-stress that matches an empirical correlation of a flat plate boundary-layer.

The aim of this exercise is to demonstrate that the error of the flow dependent variables vanishes when the grid size and time step tend to zero and to establish the asymptotic order of convergence of the method, which is supposed to match the theoretical order of the discretization technique adopted.

Solution Verification and Validation

Two typical geometries of offshore applications have been selected for the Solution Verification and Validation exercises: a smooth circular cylinder (**CASE-II**) and a straked-riser (**CASE-III**). Naturally, the selected Reynolds numbers lead to vortex shedding and so the proposed flows are periodic. Although both exercises are proposed for the two geometries, the goals of each test-case are different:

- The aim of CASE-II is mainly to check the consistency between the numerical solutions obtained with different but formally equivalent approaches.
- CASE-III is a practical test-case where the Validation procedure proposed in [4] can be tested. This will require predictions and experiments with their respective uncertainties. However, the experimental data will not be made public before the Conference.

Naturally, numerical solutions of these periodic flows will be affected by round-off, iterative and discretization errors. However, the focus of this exercise is discretization errors and so it is mandatory that round-off and iterative error contributions to the numerical error are reduced to negligible levels when compared to the discretization error. Furthermore, in order to enable a reliable estimate of the numerical uncertainty, several different calculations (grid sizes and time steps) are required per test-case.

Obviously, the proposed exercise requires a significant effort from the participants due to the large number of calculations involved and to the strict checking of the iterative convergence achieved. However, this type of exercise is essential for assessment of the capabilities available and for the identification of the challenges that are faced by mathematical modelling of complex turbulent flows.

PROPOSED TEST-CASES
General Considerations

It must be emphasized that the goal of this Workshop is to obtain numerical predictions with their respective error/uncertainties. Therefore, results are required for at least 3 different grids and 3 different time steps [20] with a minimum of 6 data points (of the possible 9). Furthermore, grid refinement must be performed in all directions simultaneously [21] and ideally the grids should be geometrically similar [22] (constant grid refinement ratio for the complete grid).

As mentioned above, numerical errors/uncertainties are not exclusively dependent on discretization error [1, 2]. However, round-off and iterative errors should be reducible to negligible levels when compared to the discretization error. In complex turbulent flows, this usually requires the use of 15 digits precision and iterative convergence criteria that ensure an iterative error at least two orders of magnitude smaller than the discretization error [23, 24]. It must be emphasized that iterative errors can be one to two orders of magnitude larger than the normalized residuals or differences between iterations of the last iteration performed [23, 24]. Therefore, iterative convergence criteria must be carefully selected to ensure that numerical uncertainties are mainly due to discretization errors.

The three test-cases proposed for this exercise are periodic flows. This means that we have two types of iterative convergence criteria involved:

- Related to the number of periods that have to be calculated to obtain a “numerically periodic” solution.
- Related to the convergence of the flow field at each time step.

Both criteria have to be checked to avoid misleading conclusions about the real magnitude of iterative errors. Iterative convergence criteria depend on the grid and time resolution (discretization error) and so it is impossible to establish “standard criteria” for all calculations. However, it must be demonstrated that the adopted criteria are sufficient to guarantee a negligible influence of iterative errors to the numerical uncertainty.

CASE-I: 3D Manufactured solution of an unsteady turbulent flow

Computational domain The computational domain is a square prism with length $0.9L_{ref}$, height $0.4L_{ref}$ and width L_{ref}. An alternative domain may be used with half the width, $0.5L_{ref}$. Figure 1 illustrates the computational domain. The computational domain does not change with time t.

$$\begin{array}{ll} 0.1L_{ref} & \leq x \leq L_{ref} \\ 0 & \leq y \leq 0.4L_{ref} \\ 0 & \leq z \leq L_{ref} \quad \text{or} \quad 0 \leq z \leq 0.5L_{ref} \end{array}$$

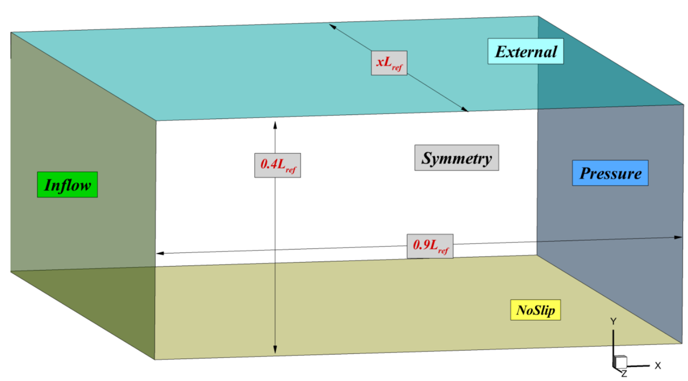

FIGURE 1. Domain and boundary conditions for MMS test-case (CASE-I).

Boundary conditions There is a lot of flexibility in the specification of boundary conditions for a manufactured solution. However, it is convenient to keep the exercise as close as possible to a realistic application. Therefore, the following boundary conditions must be applied in the calculation of this flow (see Figure 1):

- Inlet boundary ($x = 0.1L_{ref}$): prescribed velocity components and turbulence quantities.
- Outlet boundary ($x = L_{ref}$): prescribed pressure coefficient.
- Bottom boundary ($y = 0$): no-slip condition for velocity components and zero normal pressure derivative (exactly satisfied by the manufactured solution).
- Lateral boundaries ($z = 0$ and $z = 0.5L_{ref}$ or $z = L_{ref}$): symmetry conditions.
- Top boundary ($y = 0.4L_{ref}$): conditions to be imposed are left free to the participants.

Flow conditions The manufactured solution mimics a periodic near-wall turbulent flow with a Reynolds number based on the reference length, L_{ref}, and the reference velocity, V_{ref}, of $Re = 10^7$. The Strouhal number, St, of the proposed solution is equal to 1. The "wall" is the bottom boundary ($y = 0$) and at $y = 0.4L_{ref}$ the axial velocity component V_x tends to V_{ref} and the transverse velocity component V_z goes to zero.

The proposed manufactured solution satisfies mass conservation. Source functions to correct the imbalance caused by the arbitrary choice of the manufactured solution are available for the momentum equations and for the turbulence quantities transport equations of the following eddy-viscosity models:

- One-equation model of Spalart & Allmaras [25].
- One-equation model of Menter [26].
- $\sqrt{k}L$ one-equation model [27].
- Wilcox [28], TNT [29], BSL and SST $k-\omega$ [30] two-equation models.
- $k-\sqrt{k}L$ two-equation model [27].

Ideally, the exercise should be performed with all equations active, i.e. the turbulence model should be also included. However, this requires the use of an extra source term in the turbulence quantities transport equations, which is not available in many codes. Therefore, a simplest alternative is to verify only the mean flow equations, i.e. mass conservation and momentum balance, using the manufactured eddy-viscosity as a prescribed quantity.

There are also functions available with the exact solution of all dependent variables of the problem:

- Mean axial velocity component, $V_x(x,y,z,t)$.
- Mean vertical velocity component, $V_y(x,y,z,t)$.
- Mean transverse velocity component, $V_z(x,y,z,t)$.
- Mean pressure coefficient, $C_p(x,y,z,t)$.
- Eddy-viscosity, $\nu_t(x,y,z,t)$.
- Dependent variable of the Spalart & Allmaras [25] and Menter [26] one-equation models, $\tilde{\nu}(x,y,z,t)$.
- $\sqrt{k}L \equiv \Phi$ variable of the $\sqrt{k}L$ and $k-\sqrt{k}L$ one and two-equation models [27], $\Phi(x,y,z,t)$.
- Turbulence kinetic energy, $k(x,y,z,t)$.
- Turbulence frequency, $\omega(x,y,z,t)$.

All available functions use dimensionless variables with reference quantities obtained from ρ, L_{ref} and V_{ref}. The pressure coefficient C_p includes the contribution of $2/3k$.

Data to be supplied For all the dependent variables of the flow, i.e. three mean velocity components, pressure coefficient, and dependent variables of the turbulence model, at 5 different times, $t_j = 0.125T, 0.25T, 0.5T, 0.75T, T$:

- L_∞ norm of the error.

$$L_\infty(\phi) = \max\left(\phi(x_i,y_i,z_i,t_j) - \phi_{MS}(x_i,y_i,z_i,t_j)\right) \text{ for } 1 \leq i \leq N_{vol}.$$

- Root Mean Square (RMS) norm of the error.

$$\mathrm{RMS}(\phi) = \sqrt{\frac{\sum_{i=1}^{N_{vol}}\left(\phi(x_i,y_i,z_i,t_j) - \phi_{MS}(x_i,y_i,z_i,t_j)\right)^2}{N_{vol}}}.$$

ϕ stands for any of the dependent variables of the problem, ϕ_{MS} is the exact solution and N_{vol} is the number of cells of the grid. The last flow quantity requested is the "friction resistance" coefficient of the bottom boundary defined by

$$C_F = \frac{\int_{x_{min}}^{x_{max}}\int_{z_{min}}^{z_{max}} 2\nu\left(\frac{\partial V_x}{\partial y}\right)_{y=0} \mathrm{d}x\mathrm{d}z}{V_{ref}^2\,(x_{max}-x_{min})\,(z_{max}-z_{min})},$$

at the same time instants of the error norms.

CASE-II: Smooth fixed cylinder

Computational domain The classical stationary in uniform inflow smooth fixed circular cylinder is considered, see for instance [31]. This is a canonical and benchmark test-case used specially in the aerospace and offshore industry, for which blunt-body flows are very common. The scattered results of experiments and calculations for this problem (see for instance [17]) indicate that this is one of the most difficult problems to solve with the current CFD methods. It is nevertheless an easy geometry to be meshed and therefore an appropriate test-case for a solution verification exercise. Figure 2 shows the computational domain to be used and associated dimensions relative to the diameter of the cylinder. These are maybe not the optimal dimensions for a comparison with some available experimental data, but we emphasize again that the objective of this exercise is Solution Verification and not Validation. Nevertheless, the blockage effects for these dimensions are known to be small. The participants are free to choose to perform the calculations in 2D or 3D mode, and thus also to choose the width of the domain or cylinder length $L = xD$.

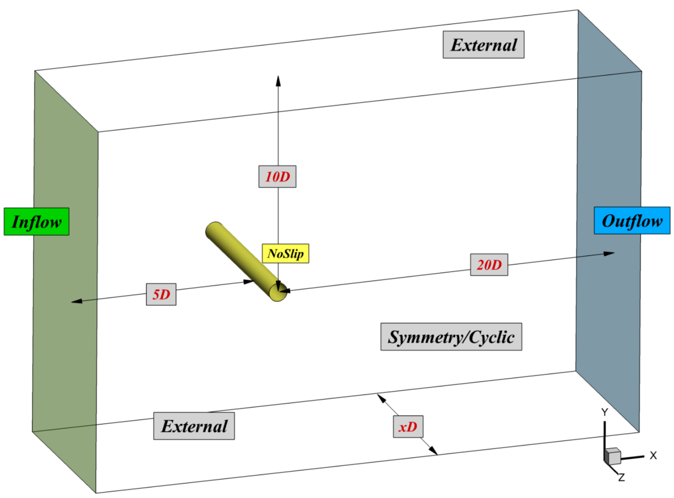

FIGURE 2. Domain and boundary conditions for cylinder test-case (CASE-II).

Boundary conditions The following boundary conditions must be applied in the calculation of this flow (see Figure 2):

- Inlet boundary: prescribed velocity components and turbulence quantities.
- Outlet boundary: zero normal gradients for all quantities.
- Cylinder boundary: no-slip condition for velocity components and zero normal pressure gradient.
- Lateral boundaries: symmetry or cyclic conditions.
- Bottom/Top boundaries: conditions to be imposed are left free to the participants.

Flow conditions For flow conditions, 5 cylinder diameter based Reynolds numbers Re_D are to be computed: $Re_D = 1\times10^3$, $Re_D = 1\times10^4$, $Re_D = 1\times10^5$, $Re_D = 5\times10^5$ and $Re_D = 1\times10^6$, see Figure 3. This is the maximum number of conditions, and the participants have to perform at least $Re_D = 1\times10^5$ and $Re_D = 5\times10^5$ conditions, close to the drag-crisis regime. All calculations should be unsteady and consider turbulence models, and as stated above, can be either 2D or 3D calculations.

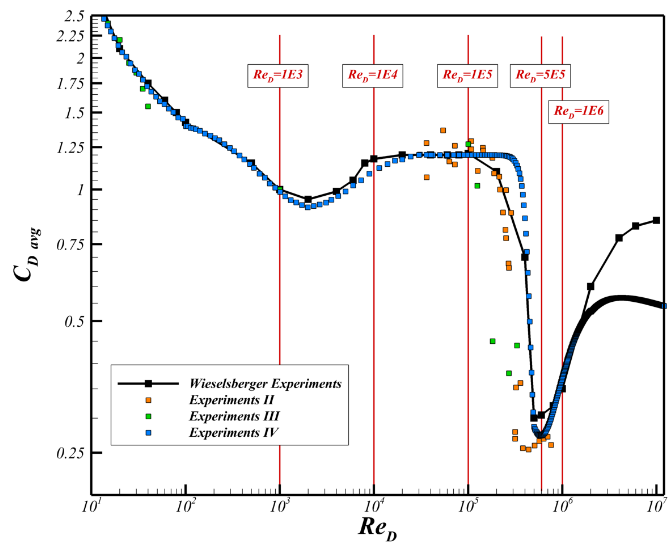

FIGURE 3. Experimental $C_{D_{avg}}$ vs Re_D curve together with proposed workshop conditions (CASE-II). From [17] and [32].

Data to be supplied Since we are dealing with a Solution Verification exercise mainly integral quantities or local quantities are considered. These are:

- Maximum, average and amplitude of drag coefficient, $C_{D_{max}}$, $C_{D_{avg}}$, $C_{D_{rms}}$, ΔC_D.
- Maximum, average and amplitude of lift coefficient, $C_{L_{max}}$, $C_{L_{avg}}$, $C_{L_{rms}}$, ΔC_L.
- Maximum, average and amplitude of base pressure coefficient C_{p_b}, being the pressure coefficient at the $\theta = 0^\circ$ location downstream of the cylinder.
- Strouhal number St based on T_{ref} corresponding to the vortex-shedding frequency calculated from the lift-coefficient first harmonic.
- Maximum, average and amplitude of separation point θ_{sep} (in 2D) or of separation vector components θ_{sep_z} (in 3D).

CASE-III: 3D straked-riser

Computational Settings Figure 4 illustrates the geometry of the straked riser chosen for this Solution Verification and Validation exercise. This geometry has been tested in MARIN's high-speed basin (HT) in 2001, and the available experimental data are currently being made public [33].

FIGURE 4. Riser geometry (CASE-III). From [33].

The geometry will be made available to the participants by means of a IGES file, but the experimental data not. The test campaign was done by towing the submersed riser in the HT basin. The associated diameter-based Reynolds number is $Re_D = 5.09 \times 10^5$. No waves have been observed during the experiments. Figure 5 shows the proposed computational domain which takes into account the distance of the riser to the free-surface and to the bottom of the basin. The domain is shortened in the axial direction in order to save computational resources (mesh resolution). The axial length of the domain is nevertheless enough not to influence the numerical results significantly. The diameter of the riser without strakes is $D = 0.2m$, and the length $L = 3.523m$.

Boundary conditions The following boundary conditions should be applied in the calculation of this flow:

- Inlet boundary: prescribed velocity components and turbulence quantities.
- Outlet boundary: zero normal gradients for all quantities.
- Riser boundary: no-slip condition for velocity components and zero normal pressure gradient.
- Lateral boundaries: symmetry or cyclic conditions.
- Bottom boundary: fixed-slip condition for velocity with $V_x = V_{ref}$ and zero normal pressure gradient.
- Top boundary: conditions to be imposed are left free to the participants.

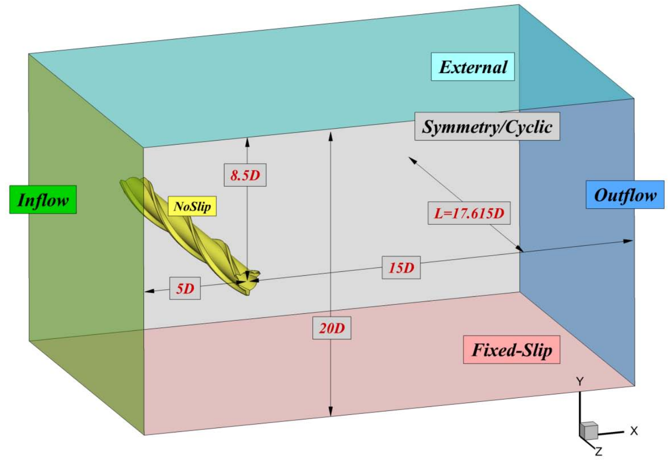

FIGURE 5. Domain and boundary conditions for riser test-case (CASE-III).

Data to be supplied For the the Solution Verification and Validation exercise the following quantities are asked:

- Maximum, average and amplitude of drag coefficient, $C_{D_{max}}$, $C_{D_{avg}}$, ΔC_D.
- Maximum, average and amplitude of lift coefficient, $C_{L_{max}}$, $C_{L_{avg}}$, ΔC_L.
- Strouhal number St based on T_{ref} corresponding to the vortex-shedding frequency calculated from the lift-coefficient first harmonic.
- C_L and C_D time history for relevant time-steps/cycles.

GENERAL CONDITIONS AND DEADLINES

The contributions to these exercises are composed of two parts:

1. A paper for a special V&V Workshop OMAE2012 session including results for *at least* one of the proposed test-cases. If all cases here proposed would be done, the results could be presented in more than one paper.
2. A reply to a questionnaire provided by the organizers and data sheets including the requested flow quantities for at least 6 different calculations using at least 3 grids and 3 time steps.

The paper will provide information about the calculations, and solution techniques used, whereas the questionnaire and data sheets will facilitate the comparison of the results of all participants. Naturally, solutions provided in a single grid with a single time step are useless for the present exercise. The papers should be submitted in the standard framework of the OMAE2012 Conference. However, data sheets and the questionnaire for the comparisons to be presented at the Workshop can be delivered to the organizers until the end of April 2012.

UNCERTAINTY ESTIMATION PROCEDURE

The reason to ask 6 results for each flow quantity ϕ is to enable the estimation of the numerical uncertainty $U(\phi)$. Assuming that the contribution of round-off and iterative errors are negligible when compared to the discretization error, it is possible to estimate $U(\phi)$ multiplying the absolute value of an estimated error $e(\phi)$ by a safety factor F_s [1,2],

$$U(\phi) = F_s e(\phi)\ . \tag{1}$$

Although a complete description of the procedure to estimate $e(\phi)$ (and define F_s) is out of the scope of this paper, we give a brief description below.

The discretization error is expressed as

$$e(\phi) = \phi_o - \phi_i \simeq \alpha_x h_i^{p_x} + \alpha_t \tau_i^{p_t} \tag{2}$$

where ϕ_i stands for any integral or local flow quantity, ϕ_o is the estimate of the exact solution, h_i is the typical cell size, τ_i is the time step, α_x and α_t are constants and p_x and p_t are the observed orders of accuracy of the space and time discretizations, respectively.

Equation (2) is solved in the least squares sense (6 data points for 5 unknowns) and the definition of F_s depends on the values of p_x and p_t and on the standard deviation of the fit. If the values of the observed order of accuracy are not reliable (or impossible to determine), alternative fits are performed with fixed values of p_x and p_t. The value of $e(\phi)$ is obtained from the fit with the smallest standard deviation.

Naturally, alternative uncertainty estimation procedures may be applied by the participants. However, we will try to apply the procedure mentioned above to all submissions.

FINAL REMARKS

The proposed Workshop is a starting point to improve the knowledge of the capabilities and limitations of “standard” CFD tools in offshore applications. Although we are addressing essentially URANS solvers, the numerical problems are common to more sophisticated mathematical models of turbulent flows, where the consequences of the numerical noise may be even less understood and worst than in URANS simulations.

Although the proposed geometries and flow conditions can be classified as “simple test-cases”, the requirements for a reliable estimation of the numerical uncertainty make any of the proposed test-cases more time consuming than what can be foreseen from “practical calculations”. However, separating numerical and modelling errors is essential for the credibility of CFD and for the improvement of the simulation tools.

The solution of a given mathematical model for a given problem is independent of the numerical technique (discretization schemes) selected. Therefore, what we would like to see at the OMAE2012 conference is consistency (overlap of error bars) between the solutions of different groups calculating the same flow with equivalent mathematical models, even if the solution shows discrepancies to the experimental data. The goal of the exercise is not to find “the numerical settings” that make the simulation match a known experiment.

A final encouragement to the potential participants: *whatever the outcome of the exercises, you will learn a lot about the behaviour, the reliability, and the quality of your computational tool.*

REFERENCES

[1] Roache, P. J., 1998. *Verification and Validation in Computational Science and Engineering*. Hermosa Publishers, Albuquerque, New Mexico.

[2] Roache, P. J., 2009. *Fundamentals of Verification and Validation*. Hermosa Publishers, Albuquerque, New Mexico.

[3] Oberkampf, W. L., and Roy, C. J., 2010. *Verification and Validation in Scientific Computing* . Cambridge University Press, Cambridge, England.

[4] American Society of Mechanical Engineers, 2008. ASME Guide on Verification and Validation in Computational Fluid Dynamics and Heat Transfer. Tech. Rep. ASME Committee PTC-61, ANSI Standard V&V-20.

[5] American Society of Mechanical Engineers, 2006. ASME Guide on Verification and Validation in Computational Solid Mechanics. Tech. Rep. ASME Committee PTC-60, ASME V&V-10-2006.

[6] American Institute For Aeronautics And Astronautics, 1998. Guide for Verification and Validation of Computational Fluid Dynamics Simulations. Tech. Rep. AIAA-G-077-1998, AIAA.

[7] American Society of Civil Engineers, 2009. Verification and Validation of 3D Free-Surface Flow Models. Tech. rep.

[8] International Towing Tank Conference (ITTC), 2002. CFD General Uncertainty Analysis in CFD Verification and Validation Methodology and Procedures. Tech. Rep. ITTC - Quality Manual 7.5-03, ITTC.

[9] Laflin Kelly, R., 2006. “AIAA CFD Drag Prediction Workshop: an Overview”. In ICAS, 25th International Congress of the Aeronautical Sciences.

[10] Vassberg, J. C., Tinoco, E. N., Mani, M., Rider, B., Zickuhr, T., Levy, D. W., Brodersenk, O. P., Eisfeldk, B., Crippa, S., Wahls, R. A., Morrison, J. H., Mavriplis, D. J., and Murayama, M., 2010. “Summary of the Fourth AIAA CFD Drag Prediction Workshop”. *In the Proceedings of 40th Fluid Dynamics Conference and Exhibit, Chicago, Illinois, USA*(AIAA Paper 2010-4547).

[11] Eça, L., and Hoekstra, M. "Proceedings of Workshop on CFD Uncertainty Analysis".
[12] Eça, L., and Hoekstra, M. "Proceedings of 2nd Workshop on CFD Uncertainty Analysis".
[13] Eça, L., and Hoekstra, M. "Proceedings of 3rd Workshop on CFD Uncertainty Analysis".
[14] Eça, L., Hoekstra, M., and Roache, P., 2005. "Verification of Calculations: an Overview of the Lisbon Workshop". *In the Proceedings of 17th AIAA Computational Fluid Dynamics Conference, Toronto, Canada*(AIAA Paper 2005-4728).
[15] Eça, L., Hoekstra, M., and Roache, P., 2007. "Verification of Calculations: an Overview of the 2nd Lisbon Workshop". *In the Proceedings of 18th AIAA Computational Fluid Dynamics Conference, Miami, U.S.A.*(AIAA Paper 2007-489).
[16] Eça, L., Hoekstra, M., Roache, P., and Coleman, H., 2009. "Code Verification, Solution Verification and Validation: an Overview of the 3rd Lisbon Workshop". *In the Proceedings of 19th AIAA Computational Fluid Dynamics Conference, San Antonio, Texas, U.S.A.*(AIAA Paper 2009-3647).
[17] Rosetti, G., Vaz, G., Fujarra, A., and Klaij, C., 2012. "Verification and Validation of URANS Calculations of the Flow Around Fixed-Smooth Cylinders for a Wide Range of Reynolds". *Journal of Fluid and Structures*. *to-be-submitted*.
[18] Roache, P. J., 2002. "Code Verification by the Method of the Manufactured Solutions". *ASME, Journal of Fluids Engineering*, ***124***, p. 410.
[19] Eça, L., Hoekstra, M., Hay, A., and Pelletier, D., 2007. "On the construction of manufactured solutions for one and two-equation eddy-viscosity models". *International Journal for Numerical Methods in Fluids*, ***54***, p. 119154.
[20] Eça, L., Vaz, G., and Hoekstra, M., 2010. "Code Verification, Solution Verification and Validation in RANS Solvers". *In Proceedings of OMAE2010, Shangai , China*, June.
[21] Salas, M. D., 2006. "Some Observations on Grid Convergence". *Computers and Fluids*, ***35***, p. 688192.
[22] Eça, L., and Hoekstra, M., 2009. "Error Estimation Based on Grid Refinement Studies: a Challenge for Grid Generation". *In the Proceedings of Conferência de Métodos Numéricos en Inginería 2009, SEMNI Barcelona, Spain*(AIAA Paper 2010-4547).
[23] Eça, L., and Hoekstra, M., 2006. "On the Influence of the Iterative Error in the Numerical Uncertainty of Ship Viscous Flow Calculations". *In the Proceedings of 26th Symposium on Naval Hydrodynamics, Roma, Italy*.
[24] Eça, L., and Hoekstra, M., 2009. "Evaluation of numerical error estimation based on grid refinement studies with the method of the manufactured solutions". *Computers and Fluids*, ***38***, p. 15801591.
[25] Spalart, P., and Allmaras, S., 1992. "A One-Equation Turbulence Model for Aerodynamic Flows". *In the Proceedings of AIAA 30th Aerospace Sciences Meeting, Reno, Nevada, USA*(AIAA92-0439).
[26] Menter, F. R., 1997. "Eddy-Viscosity Transport Equations and their Relation to the $k-\varepsilon$ Model". *ASME, Journal of Fluids Engineering*, ***119***, p. 876884.
[27] Menter, F., Egorov, Y., and Rusch, D., 2005. "Steady and Unsteady Fow Modelling using $k-\sqrt{k}L$ Model". *Turbulence, Heat and Mass Transfer*, ***5***.
[28] Wilcox, D. C., 1998. *Turbulence Modeling for CFD*, 2nd ed. DCW Industries.
[29] Kok, J., 1999. Resolving the Dependence on Free-stream values for the $k-\omega$ Turbulence Model. Tech. Rep. NLR-TP-99295, NLR, National Aerospace Laboratory.
[30] Menter, F., 1994. "Two-equation Eddy Viscosity Turbulence Models for Engineering Applications". *AIAA Journal*, ***32***, pp. 1598–1605.
[31] Williamson, C., and Govardhan, R., 2004. "Vortex-Induced Vibrations". *Anual Review of Fluid Mechanics*(36), pp. 413–455.
[32] Wilde, J., 2012. "Lift Forces on Oscillating Circular Cylinders at Critical and Super-Critical Reynolds Numbers". *Journal of Fluids & Structures*. (*to-be-submitted*).
[33] Wilde, J., 2012. "Experiments for High Reynolds Numbers VIV on Straked Risers". *Journal of Fluids & Structures*. (*to-be-submitted*).