Workshop on CFD Uncertainty Analysis
Lisbon, Portugal, 21-22 October 2004

Examining the ERCOFTAC C-18 Test Case

Zlatko Zlatev[1]
Bulgarian Ship Hydrodynamics Centre, 9003 Varna, Bulgaria

ABSTRACT

The paper presents some results from computations of the incompressible viscous high Re-number flow over a single hill (test case C-18 of the ERCOFTAC experimental database) by means of the commercial CFD code FLUENT 6.1. A predefined set of eleven geometrically similar computational grids, together with inlet velocity, pressure and turbulence profiles, supplied preliminarily by the Workshop organizers, were used in the computations. Assessment of uncertainty of the CFD results, based on grid triplets, is presented.

INTRODUCTION

The geometry of the computational domain is shown in Fig. 1. According to [1], the fluid is water at normal pressure and temperature, with no heat exchange through the walls; the flow is supposed to be two-dimensional and steady. The height of the hill h = 0.028 m is the reference length, and the reference velocity is $\mathbf{U}_{REF}$ = 2.147 m/s. The Reynolds number, based on these parameters and conditions, is about 6.10^5.

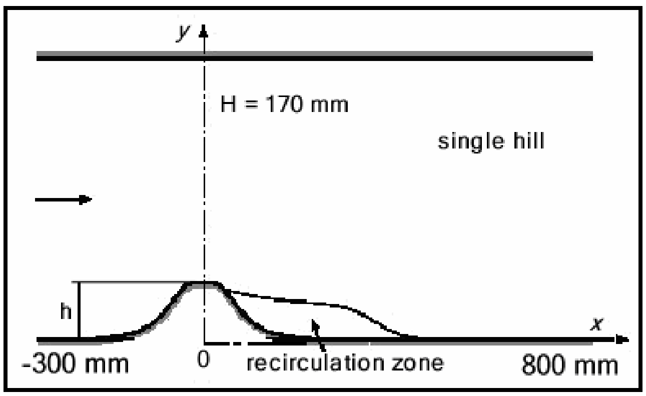

Figure 1. *Domain geometry*

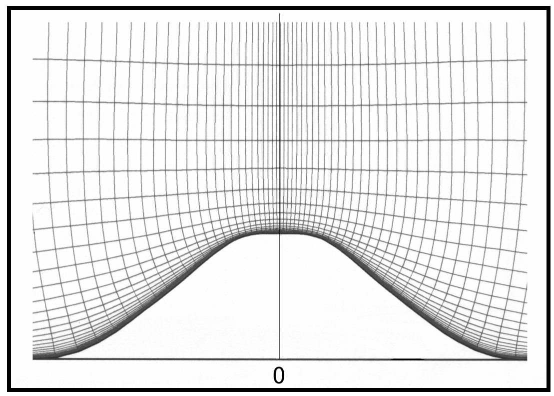

Figure 2. *A grid fragment (C-18, Set A)*

The CFD analysis code FLUENT is a finite volume solver based on the RANS equations in their strong conservative form, capable of handling both compressible and incompressible laminar/turbulent flows in zones of fixed or variable geometry, plus additional equations (if any) for chemical reactions, phase changes, particle flows, etc. Structured, unstructured or hybrid (multiblock; stationary, moving or deforming) grids can be used.

COMPUTATIONAL CONDITIONS

Discretization schemes

Standard (first order) discretization schemes are used for the continuity and momentum equations to start the computations, followed by switching to second-order upwinding after settling of the residuals, combined with the SIMPLE pressure-velocity coupling scheme. The same sequence is used for the turbulence quantity.

Choice of a turbulence model

The one-equation model of Spalart & Allmaras (1992) and its "strain-vorticity production" modification by Dacles-Mariani et al. (1995), both embedded in the code, were tried yielding no substantial differences in the results. The results presented herewith are based on the standard S-A low-Re number version. A description

[1] z.zlatev@bshc.bg

of both models can be found in their respective references [2], [3].

Boundary conditions

Inflow:

$\mathbf{U^1}$(y/h) = $\mathbf{U_{REF}} * \bar{\mathbf{U}}^1$(y/h), where $\bar{\mathbf{U}}^1$(y/h) is the non-dimensional profile for the relevant grid case;

$\mathbf{U^2}$(y/h) = const = 0.

$\boldsymbol{p}_{\mathbf{gauge}}$ (y/h) = 0, relative to some reference pressure $\boldsymbol{p}_{\mathbf{REF}}$.

For the S-A model (modified turbulent kinematic viscosity):

$\tilde{\nu}$ (y/h) = $\bar{\tilde{\nu}}$ (y/h) *$\mathbf{U_{REF}}$*h, where $\bar{\tilde{\nu}}$ (y/h) is the non-dimensional profile for the relevant grid case.

Outflow:

- zero diffusion flux for all flow variables,
- an overall mass balance correction.

This means that the conditions of the outflow plane are *extrapolated from within the domain* and have *no* impact on the upstream flow. The outflow velocity and pressure are updated in a manner consistent with a fully-developed flow assumption (when there is no area change at the outflow boundary) [4].

Wall(s):

No-slip, stationary, non-permeable, adiabatic wall ($\mathbf{U^1} = \mathbf{U^2} = 0$), hydraulically smooth; zero normal gradient for pressure; $\bar{\tilde{\nu}}\,|_{\text{wall}} = 0$ – set internally by the program.

Computational grids

The chosen set (Set A) consists of eleven two-dimensional, single-block, structured grids (ranging from 101x101 to 401x401 nodes, i.e. a max grid refinement ratio of 4; a fragment of the coarsest 101 x 101 grid is shown on Fig. 2). They are nearly orthogonal, with the values of the equiangle-skew ratio Q_{EAS} [5] not exceeding 0.075 for the entire grid set.

Convergence control

Convergence was controlled based on the scaled residuals [4]. All the cases were run in double precision until the residuals settled at constant values near the estimated machine DP accuracy of about 1.10^{-15}.

Uncertainty assessment method

According to [6,7], the following method is adopted: suppose we have a set of grids, all of which are in the *asymptotic* range. Based on Richardson's extrapolation, the asymptotic error can be expressed as follows:

$$\boldsymbol{\delta}_{RE} = \phi_i - \phi_0 = \alpha h_i^p , \tag{1}$$

where: ϕ_i is an arbitrary monitored value pertaining to the i-th grid of the set, ϕ_0 is an estimate of the exact solution, α is a constant, p is the *observed* order of accuracy and h_i is a representative grid cell size. At least three grids are necessary to determine the unknowns ϕ_0, α and p, and this is the approach adopted in this study. The three grids must satisfy the conditions: *a*) $(\phi_2 - \phi_1) \times (\phi_3 - \phi_2) > 0$, and *b*) $p > 0$ (ϕ_1 is the finest grid). The values of p and $\boldsymbol{\delta}_{RE}$ are obtained from the equations:

$$\frac{\phi_3 - \phi_2}{\phi_2 - \phi_1} - \left(\frac{h_2}{h_1}\right)^p \frac{\left(\frac{h_3}{h_2}\right)^p - 1}{\left(\frac{h_2}{h_1}\right)^p - 1} = 0 \ , \quad \boldsymbol{\delta}_{RE} = \frac{\phi_2 - \phi_1}{\left(\frac{h_2}{h_1}\right)^p - 1}$$

Then, in the context of the CGI method of Roache [8], the uncertainty U is estimated from the formula:

$$U = F_s\,|\boldsymbol{\delta}_{RE}| ,$$

where F_s is an empirical safety factor, taking values between 1.25 and 3 depending on the value of the observed order of accuracy p.

COMPUTATIONAL RESULTS

The whole C-18 Set A set of grids was used in the computations. Results have been obtained for two of the points of observation, namely [x = 0, y = 1.25h] and [x =2.5h, y = 0.25h], as well as for the values $(C_F)_b$ – bottom wall friction coefficient, $(C_P)_b$ - bottom pressure coefficient, $(C_F)_t$ – top wall friction coefficient, x_{sep}/h, x_{ret}/h – non-dimensional separation and reattachment locations. Regarding the third point [x = 5.357h, y = 0.107h], after having established huge discrepancies with the experimental data for the velocity profiles and separation bubble length, it was decided to abandon any further calculations. Moreover, local pressure coefficient data is not presented, since with the aforementioned outflow BC setup the pressure has slight variations along y (not fully recovered) and would have required area-weighted averaging (i.e. introducing additional error) in order to be comparable with the required BC of $p = 0$ at outflow[1].

The values of the monitored variables, together with their uncertainties obtained at the monitoring points, are shown in Tables 1 and 2. Figure 3 illustrates the grid convergence histories at the point [x = 0, y = 1.25h].

Table 1. *Results for the field quantities.*

VARIABLE	x = 0 y = 1.25h	x = 2.50h y = 0.25h
U^1	1.16917	-0.2074
Uncertainty U^1	0.00510	0.0013
U^2	0.11551	0.01464
Uncertainty U^2	0.00016	0.00120
ν_t	0.001952	0.006380
Uncertainty ν_t	0.000027	0.000004

Table 2. *Integral quantities.*

VARIABLE	VALUE
$(C_F)_b$	0.02388
Uncertainty $(C_F)_b$	0.00015
$(C_P)_b$	0.19320
Uncertainty $(C_P)_b$	0.00018
$(C_F)_t$	0.05713
Uncertainty $(C_F)_t$	0.00003
x_{sep}/h	0.2079
Uncertainty x_{sep}/h	0.0021
x_{ret}/h	8.386
Uncertainty x_{ret}/h	0.0075

The uncertainty estimates are based predominantly on the grid triplet including the 121x121, 201x201 and 361x361 grids (except the cases of oscillatory convergence, where suitable triplets from the monotonic leg of the history curves were used). Despite the fact that all of the triplets used yielded values of p between 1 and 2, alternative calculations with arbitrary combinations of grids were conducted showing a wide range of scatter for p, which just confirms the conclusion in [7] that the sole usage of grid triplets for uncertainty estimation may be rather risky in real situations. This raises the need of more sophisticated approaches to the problem, as the least-square approach, proposed by Eça & Hoekstra [6], for example.

As stated by Eça & Hoekstra in [7], there is *at least* one cause for the sensitivity of p to the choice of the grid triplets: the existence of scatter in the data. Three major sources of scatter exist: the lack of geometric similarity of the grids[2], the interpolation required in the post-processing of the numerical solution and the use of switches in the turbulence model implementations. One possible switch of the last kind is virtually avoided, as all of the grids have $y^+ < 1$ and sufficient number of wall-adjacent cell layers within the viscous region, hence a total resolution of the viscous sublayer takes place in all of the cases (no switching to wall functions; see the recommendations in [4]). Analysis of the curves shown on Fig.3 brings up the idea that equally well the scatter might come from the accuracy of integration of the input velocity profiles, in view of the high gradients near the walls. This would mean different overall fluxes, i.e. different calculated flows for the separate grid cases. Thus, in our case, the difference in volumetric flux between the first and second grid

[1] This was noticed too late to reformulate the boundary conditions according to the Workshop conventions.
[2] In this case - avoided by the special construction of the grids.

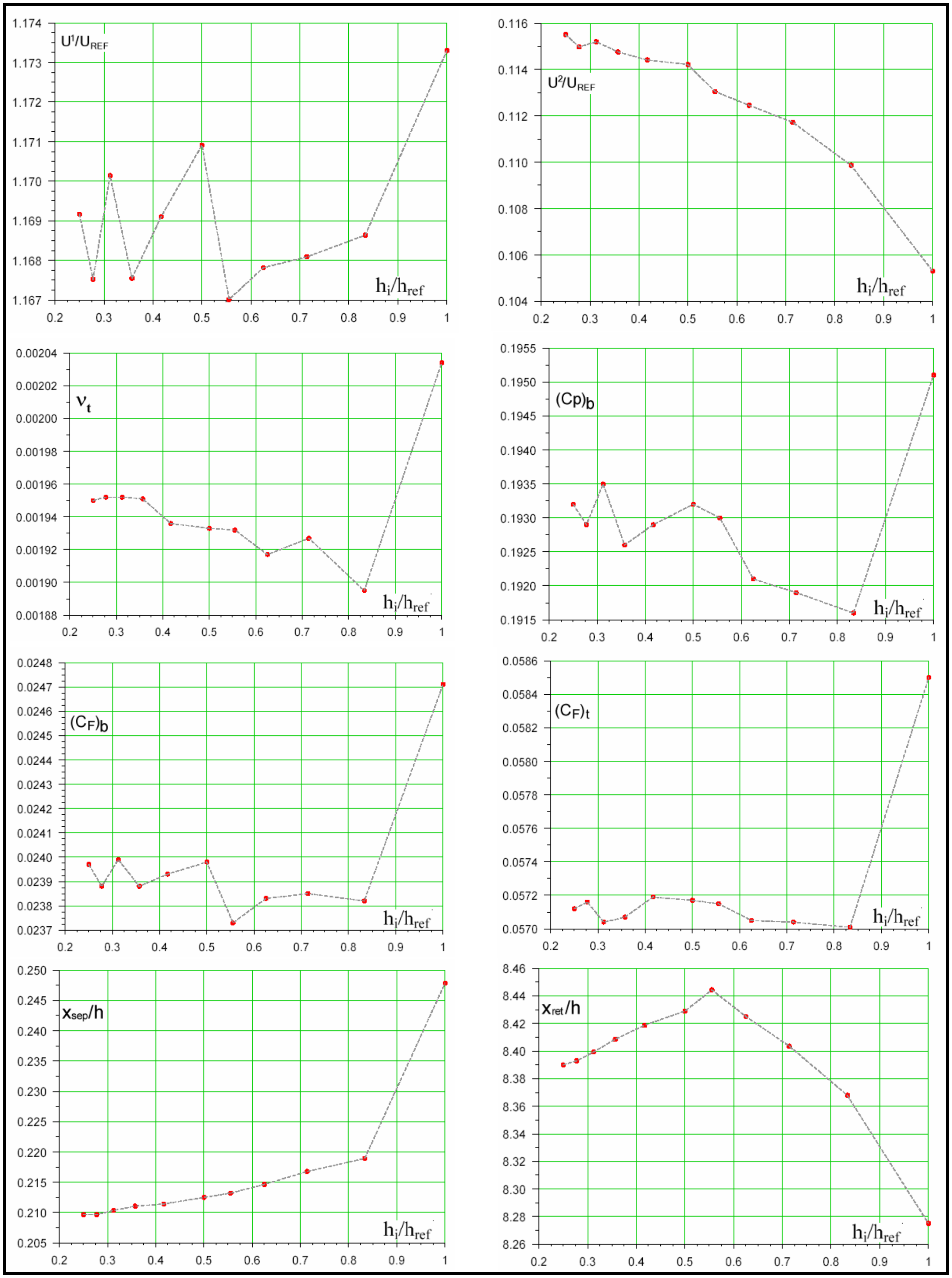

Figure 3. *Grid convergence histories of the flow variables.*

is of the order of 1.3%; however, the experiment to “scale down” the input velocity profile of the first grid to that of the second and repeating the calculation with the new profile has really shown some “flattening” of the convergence history curves at their first point.

Finally, though not a goal of this exercise, some comparison with the available test data (**U**[1] velocity transverse profiles) from [1] is presented on Fig. 4. It is evident that the coincidence is good as long as x is less than the experimentally established value of x_{ret}, and rapidly deteriorates after that, mainly due to the severe overestimation of the reattachment length. This may be considered as an expected behavior of the S-A model, since similar situation was reported in [10] for the analysis of the second half of the ERCOFTAC C-18 case – the flow over periodic hills.

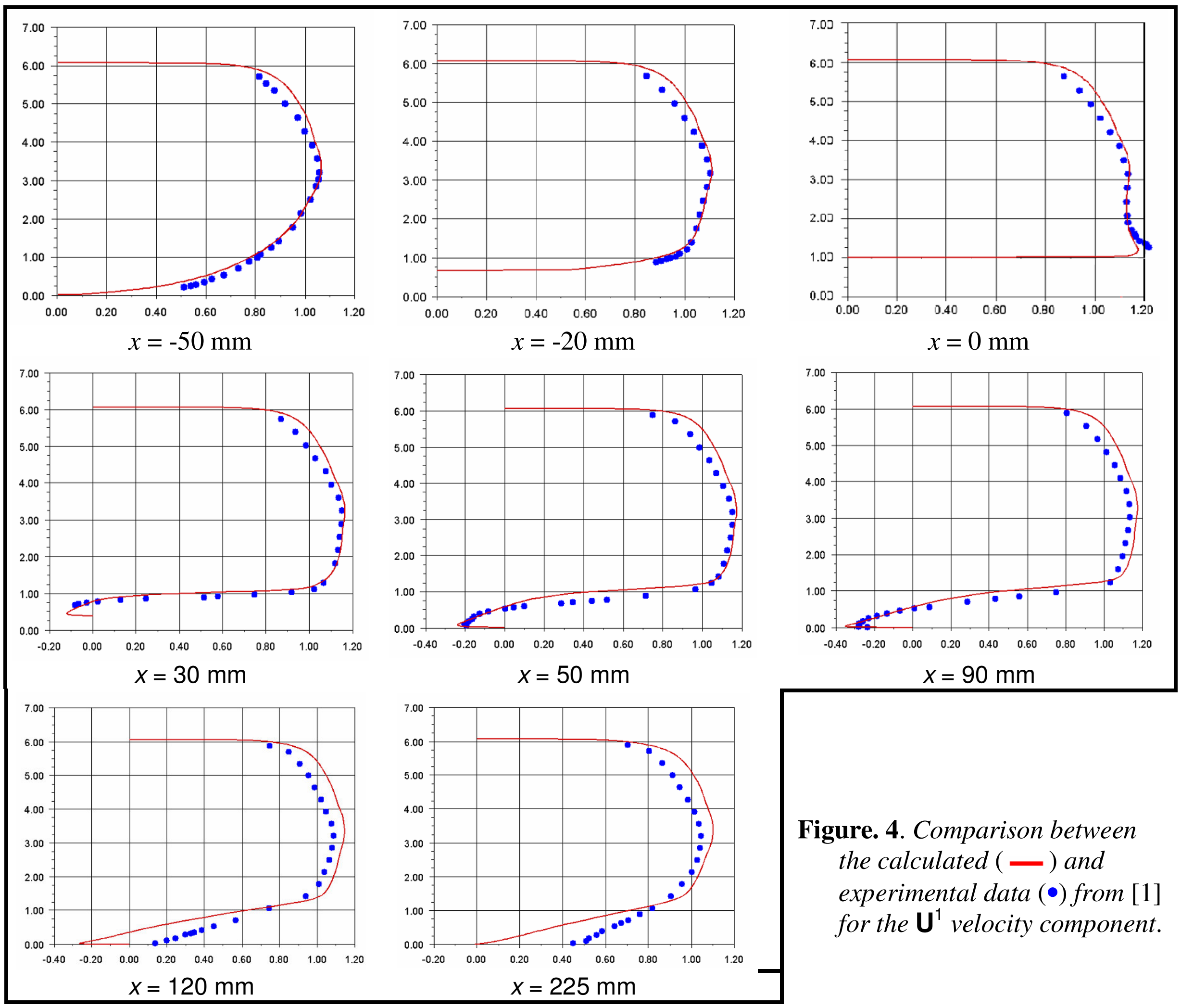

Figure. 4. *Comparison between the calculated (—) and experimental data (•) from* [1] *for the* **U**[1] *velocity component.*

CONCLUSION

Grid convergence study for the ERCOFTAC C-18 case of a flow over a single hill is carried out by means of a commercial CFD code, using a predefined set of eleven grids and the Spalart-Allmaras turbulence model. An attempt for uncertainty estimation, based on Richardson extrapolation and the GCI method, using grid triplets, was implemented. The results obtained and their analyses have shown, that in general, more sophisticated (and resource-saving) methods of uncertainty estimation are necessary in order to increase the reliability of the CFD analyses.

ACKNOWLEDGEMENT

Deepest thanks to Fluent – Europe, as well as to SimTec, Ltd. of Thessaloniki, Greece, for their super-fast response in providing me with one of the references in this paper.

REFERENCES

[1] The ERCOFTAC Classical Database (http://cfd.me.umist.ac.uk/ercoftac)

[2] Spalart, P. R., S. R. Allmaras (1992). A One-Equation Turbulence Model for Aerodynamic Flows. Technical Report AIAA-92-0439, American Institute of Aeronautics and Astronautics.

[3] Dacles-Mariani, J. , G. G. Zilliac, J. S. Chow, P. Bradshaw (1995). Numerical/Experimental Study of a Wingtip Vortex in the Near Field. *AIAA Journal*, 33(9):1561-1568.

[4] Fluent 6.1 User's Guide. *Fluent Inc., February, 2003*.

[5] Gambit 2.1 User's Guide. *Fluent Inc., February, 2003*.

[6] Eça, L., M. Hoekstra (2002). An Evaluation of Verification Procedures for CFD Applications. *24th Symp. Nav. Hydrodyn., Fukuoka, Japan*.

[7] Eça, L., M. Hoekstra (1999). On the Numerical Verification of Ship Stern Flow Calculations. 1st *MARNET-CFD Workshop, Barcelona*.

[8] Roache, P. J. (1998). Verification and Validation in Computational Science and Engineering. *Hermosa Publishers*.

[9] Eça, L., M. Hoekstra (2003). Uncertainty Estimation: A Grand Challenge for Numerical Ship Hydrodynamics. *6th NuTTS, Rome, September*.

[10] Rumsey, C. L. (2003). Effect of Turbulence Models on Two Massively-Separated Benchmark Flow Cases. *NASA/TM-2003-21241*.