COMPUTATIONAL MESH AND RANS MODEL OF TURBULENCE SENSITIVITY IN CFD OPTIMIZATION OF SLOTTED FLAP

Tomáš Koutník
Institute of Aerospace Engineering
Brno University of Technology
koutnik@fme.vutbr.cz

Petr Doupník
Institute of Aerospace Engineering
Brno University of Technology
doupnik@fme.vutbr.cz

Abstract. As powerful and relatively fast and cheap tools, CFD solvers are used for solution of wide range of problems in the aerodynamics including the search for optimal position of slotted flap. If relative difference between CFD result and real behavior of the flow varies strongly with the flap configuration, predicted optimum may vary from the real one as well. Therefore a test case focused on estimation of this type of error was conducted. This article presents results of computed 2D aerodynamic of NACA 662-216 airfoil with 25% slotted flap deployed and located at 25 different positions. Combination of unstructured and hybrid computational meshes with total number of elements ranging from 79 to 330 thousands and three RANS models of turbulence were applied. For each model of turbulence a consistency of results with respect to the mesh density is discussed in the article and compared to the general recommendations. Finally, the data obtained from CFD solution were validated against the wind tunnel measurement and conclusions were made.

Keywords. Slotted flap, CFD, wind tunnel, validation, mesh sensitivity, model of turbulence

1 Introduction

The key topic of high-lift system design for general aviation aircraft is to meet stall speed requirements of certification specifications. Therefore the designer's main focus is on the maximum lift coefficient achievable in aircraft's landing configuration, as the basic equation for stall speed computation reveals.

$$V_{S0} = \frac{2 \cdot m_{TOW} \cdot g}{\rho \cdot A \cdot c_{Lmax}}$$

Coefficient c_{Lmax} itself, beside parameters of wing-layout matter, is dependent on aerodynamic characteristics of airfoil with deployed high-lift device. The scope of presented paper is on prediction of aerodynamic characteristics of the airfoil with the slotted flap on its trailing edge, since it is one of the most common high-lift devices used in recent general aviation aircraft designs.

These characteristics can be obtained using CFD codes, as they become more affordable than wind tunnel measurements. Thus, prediction capability of these tools should be of designer's concern. Many studies were conducted to compare results from CFD against those from wind tunnel test and to discuss general recommendations for CFD solver and computational mesh settings, as summarized by Rumsey and Ying [1]. Typical validation task includes comparison of drag polar and boundary layer velocity profiles or flow field in the cove of the flap for one position of the flap corresponding to given deflection angle.

However, typical 2D optimization task in high-lift design is the search for optimal position of deflected flap, because it is an important factor affecting maximum lift coefficient. Thus, agreement of results through the design space and their consistency with mesh size and RANS model of turbulence used should be studied. This type of verification was made i.e. by Wild [2], whose paper deals with positioning of the slat, or solved in slightly different form within the EUROLIFT II project [3, 4], but still it is not the common practice.

2 Methodology

The goal of this study is to create a successful validation case comparing measured and computed aerodynamic coefficients for multi-element airfoil with different configurations of the slotted flap, so the ability of CFD tools to find the same optimal flap position as that found in the wind tunnel is tested. This evaluation is made for several mesh densities and RANS models of turbulence to define their influence on results, which are presented in form of isocontour maps.

2.1 Multi-element airfoil geometry

The main criterion for geometry selection from variety of airfoils with slotted flap was accessibility of wind tunnel measurement data for these geometries, where aerodynamic coefficients are measured for sufficient amount of flap configurations. Also with respect to previous related work at Institute of Aerospace Engineering, the NACA 66_2-216 airfoil with a 0.25-chord slotted flap was chosen. Airfoil geometry was digitized from coordinates given in related report [5] and its shape analyzed in CAD software Catia with very little discontinuities in curvature found. Figure 1 shows detail of the cove with 25 highlighted positions of the flap examined using CFD tools. These stations cover close neighborhood of the position providing the highest maximum lift coefficient possible found by measurement. Depicted isocontours of maximum lift coefficient refer to set flap deflection angle 40°, suitable for landing configuration.

CAD model of clean airfoil without the flap was also created, so CFD results for airfoil with/without flap can be compared.

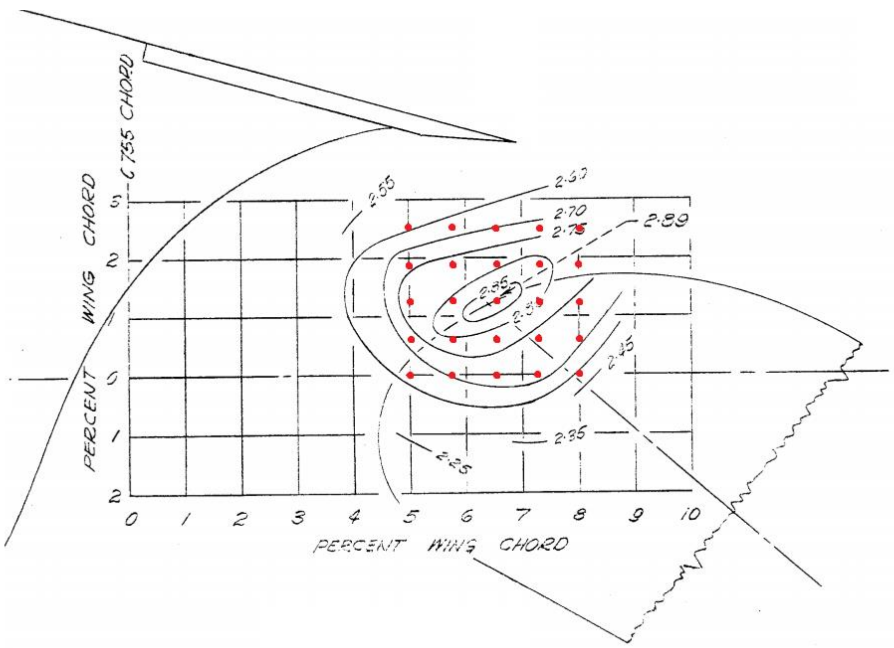

Figure 1: Contours of the maximum lift coefficient, NACA 66_2-216 with 25%c slotted flap at $\delta_f = 40°$, interpolated from measured data [5]. Red dots represent positions of the flap examined in CFD.

2.2 Computational mesh

For all cases, a round domain with diameter of 80 chord lengths was created. Airfoil itself is located 5 chord lengths below and 7 chord lengths in front of the domain's center.

Four computational meshes used were unstructured with layer of prisms on airfoil's surface and outline divided into 50 elements. Height of elements adjacent to the surface was about 0.006mm in order to maintain y+ value below 1. Also the volume of elements in the cove of the flap was limited. The round domain mentioned before filled with unstructured elements, was also used to extend formerly used structured mesh with relatively small C-shaped domain [12]. This new hybrid mesh should be able to simulate the effect of the mesh type in airfoil's nearby region. Details of each of 5 mesh sizes are listed in table 1. Approximate values of mesh size are given for unstructured meshes, because it varies slightly with the changing position of the flap.

Table 1: List of mesh sizes

Mesh	Mesh type	Max. cell volume in flap cove	Number of elements		Mesh size
			on airfoil	on flap	
a	unstructured + prisms	2,5	490	230	79000
b	unstructured + prisms	1,5	761	385	112000
c	unstructured + prisms	0,75	1585	747	187000
d	unstructured + prisms	0,5	3141	1492	330000
p	hybrid	-	974	605	246785

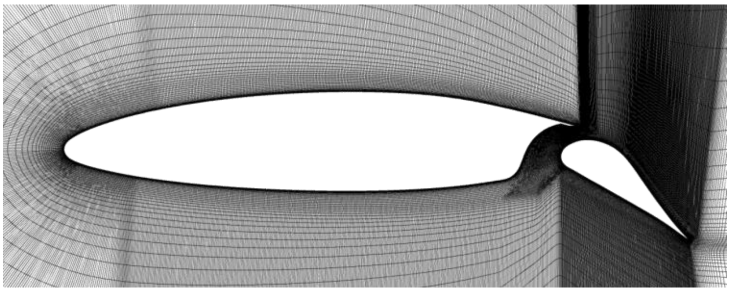

Figure 2: Structured part of mesh "p" close to the airfoil

2.3 CFD solver settings

For all computations the well-known CFD solver Ansys Fluent [6] was used. Each mesh was combined with three RANS models of turbulence – Spalart-Allmaras [7], realizable k-ε [8] and shear-stress transport k-ω [9], which are widely used in aircraft design applications. All computations were solved as in steady state because of the total number of cases. For every single flap configuration, mesh and solver settings a set of 27 angles of attack ranging from -8° to 18° was solved. Convergence process for one complete aerodynamic polar required of about 70 000 iterations, resulting in more than 26 000 000 iterations for the whole evaluation task.

A pressure far field boundary condition was set to the outline of the computational mesh with Mach number 0.19 and operating pressure 95 588 Pa in order to maintain Reynolds number 5.1e6 for which tunnel measurement was conducted [5]. Turbulence of the free stream flow was given by its intensity 0.3% and length scale 0.03m. These values could not be in exact agreement with contemporary conditions in the wind tunnel No.1 of the Ames Aeronautical laboratory, but their influence is very minor as found during preliminary runs.

3 Discussion of results

On figure 3 typical comparison of lift curves and aerodynamic polars is shown, with flap position in the center of previously defined area of interest. Used computational mesh was "d", the densest unstructured mesh. The main visible difference between measured and CFD results is in critical angle of attack and drag coefficient. As referred i.e. in [10], in case of flapless airfoils the main cause is all-turbulent boundary layer of selected RANS models of turbulence. Unfortunately, attempts to perform computations on flapped NACA 66_2-216 with model enabling boundary layer transition have encountered persisting convergence problems and were not successful so far.

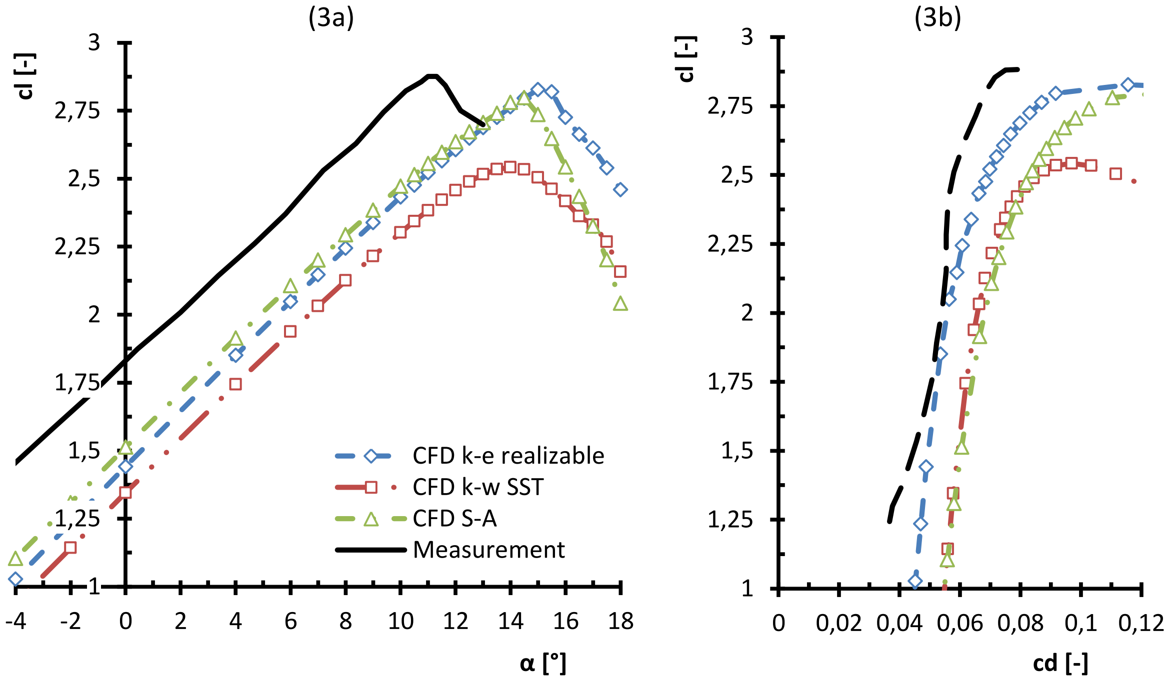

Figure 3: Comparison of results from computations for flap position x = 6.25%c, y = 1.25%c, mesh „d" with measurement [5]: (3a) lift curve (3b) aerodynamic polar

Nevertheless, especially with k-ε and Spalart-Allmaras models of turbulence a good agreement in maximum lift coefficient was found, where this value is slightly underestimated. Figures 4 and 5 show changes in predicted maximum of lift coefficient (and its error respectively) with the position of the flap. Unlike k-ε and Spalart-Allmaras, SST k-ω model predict lower $(c_l)_{max}$ then expected from measurements all over the examined area. Flap position optimal for landing is suggested to be further from that found in wind tunnel also, but none of the models reached the measured optimum exactly and according to all models a smaller gap between flap and airfoil is needed.

Also variation of results obtained from computations with meshes of different density is not crucial. Significant parameter appears to be the number of elements on flap's surface. The mesh with structured arrangement nearby the airfoil shows slightly different results distribution, but it is still comparable to those obtained with unstructured meshes.

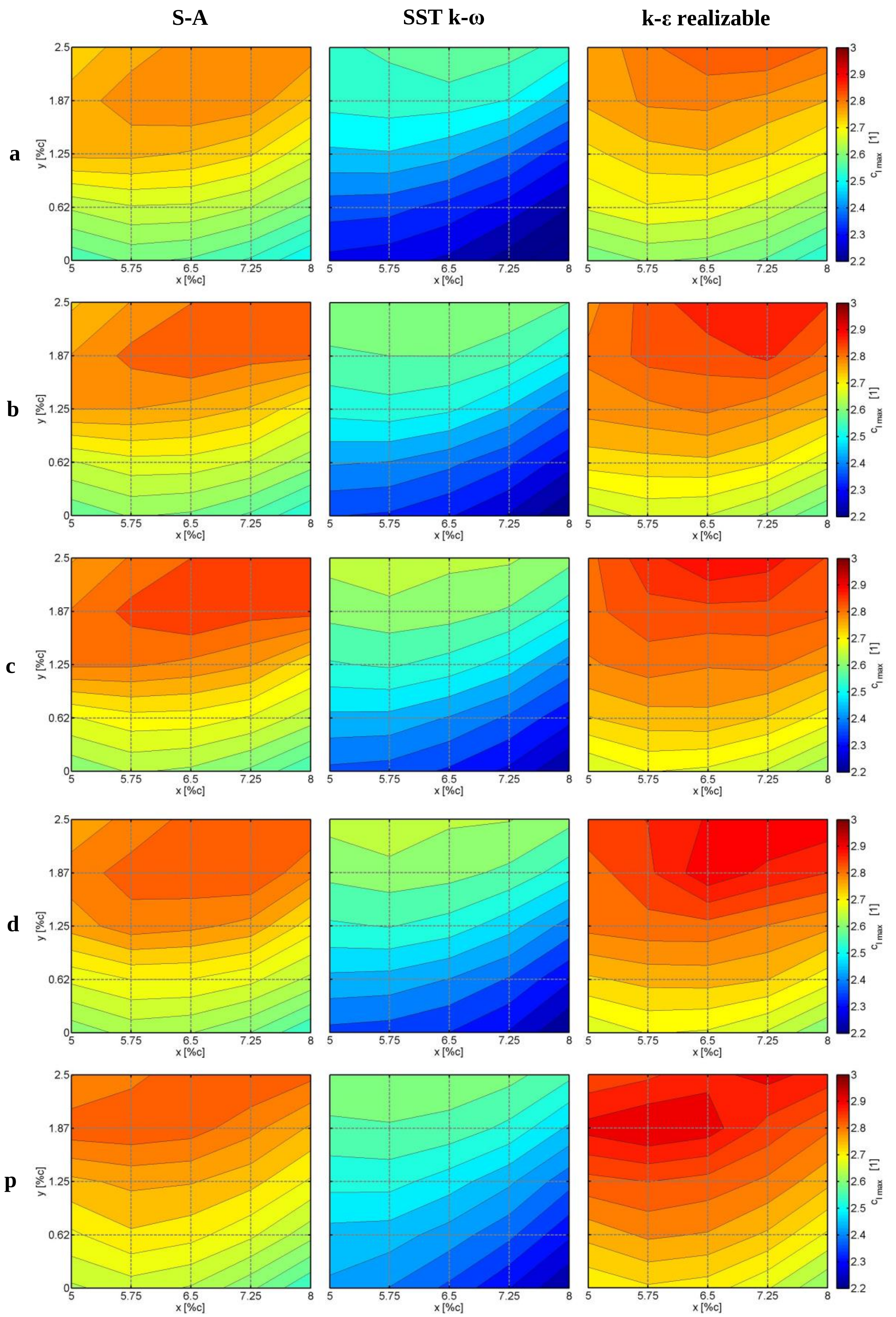

Figure 4: Contours of the maximum lift coefficient, NACA 66_2-216 with 25%c slotted flap at $\delta_f = 40°$, computed.

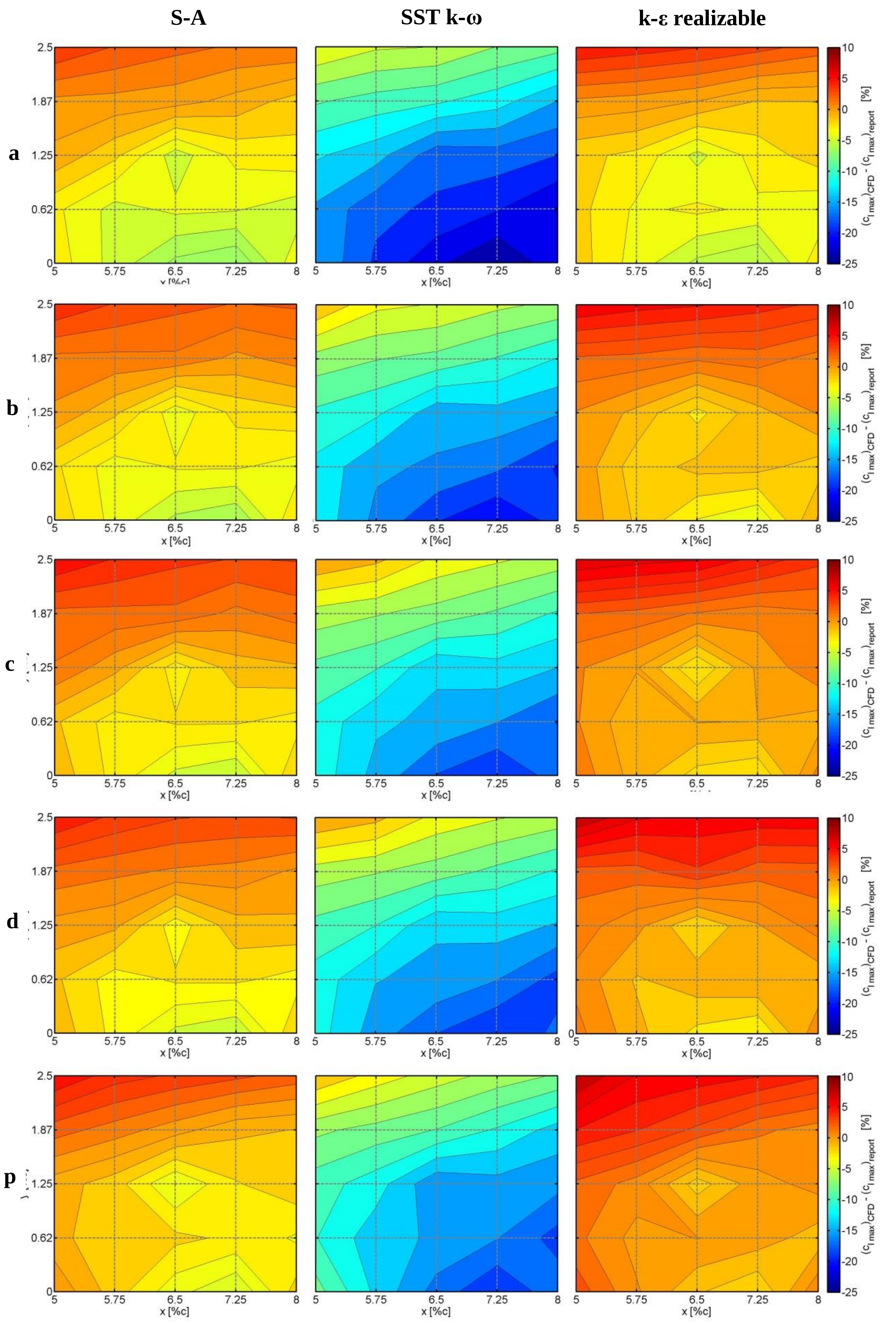

Figure 5: Contours of error in the maximum lift coefficient, NACA 66_2-216 with 25%c slotted flap at $\delta_f = 40°$

4 Conclusion

Presented comparison of measured and computed high-lift characteristics of an airfoil with slotted flap illustrates sensitivity of results to mesh density and used setting of CFD solver. Although the distribution of maximum lift coefficient showed in fig. 4 did not match the measured optimum, a clear consistency in results can be seen through variety of computational meshes. With number of elements on the flap's surface higher than 600 only negligible differences can be noted. The use of structured mesh also does not offer significant advantage in results over the unstructured one with prismatic layer.

Preferable choice of compared models of turbulence is Spalart-Allmaras. Capability of this model to predict the value of the lift coefficient is sufficient in all tested positions of the flap, and comparable to k-ε realizable model. Plus, Spalart-Allmaras is more stable at higher angles of attack, where other models tend to oscillate in coefficients.

In future work the source of disagreement in computed and measured optimal flap position should be found. One possible option is the distance of tunnel walls from the airfoil during tunnel tests, which was lesser than 1.5 chord lengths. More precise results (including in drag coefficient) could be reached by using more sophisticated models of turbulence with boundary layer transition with unsteady solution to avoid oscillation in results, which would, on the other hand, increase computational times excessively.

Dedication

These outputs were supported by the project NETME Centre, regional R&D centre built with the financial support from the Operational Programme Research and Development for Innovations within the project NETME Centre (New Technologies for Mechanical Engineering), Reg. No. CZ.1.05/2.1.00/01.0002.

References

[1] Rumsey, Ch. L., Ying, S. X.: Prediction of high lift: review of present CFD capability, Progress in Aerospace Sciences 38 (2002), pp. 145 – 180.

[2] Wild, J. : Validation of Numerical Optimization of High-Lift Multi-Element Airfoils based on Navier-Stokes-Equations, 20th Applied Aerodynamics Conference, St. Louis (USA), 24.-26.06.2002, AIAA 2002–2939, 2002.

[3] Rudnik, R., Huber, K., Melber-Wilkending, S.: EUROLIFT Test Case Description for the 2nd High Lift Prediction Workshop, 30th AIAA Applied Aerodynamics Conference, New Orleans, Louisiana (USA), 25.-28.06.2012, AIAA 2012-2924, 2012.

[4] Wild, J., Brezillon, J., Amoignon, O., Quest, J., Moens, F., Quagliarella, D.: Advanced High-Lift Design by Numerical Methods and Wind Tunnel Verification within the European Project EUROLIFT II, 25th AIAA Applied Aerodynamics Conference, Miami, FL (USA), 25.-28.06.2007, AIAA 2007-4300, 2007.

[5] Holtzclaw R. W., Weisman. Y.: Wind-tunnel investigation of the effects of slot shape and flap location on the characteristics of a low-drag airfoil equipped with a 0.25-chord slotted flap, NACA-MR-A4L28, 1944.

[6] ANSYS Fluent User's Guide, Release 15.0, ANSYS, Inc., 2013.

[7] Spalart, P., Allmaras, S., A one-equation turbulence model for aerodynamic flows. Technical Report

AIAA-92-0439. American Institute of Aeronautics and Astronautics. 1992.

[8] Shih, T.-H., Liou, W. W., Shabbir, A., Yang, Z., Zhu J. A New - Eddy-Viscosity Model for High Reynolds Number Turbulent Flows - Model Development and Validation. Computers Fluids. 24(3). 227–238. 1995.

[9] Menter, F. R.. Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications. AIAA Journal. 32(8). 1598–1605. August 1994.

[10] Hansen, T. Modeling the Performance of the Standard Cirrus Glider using Navier-Stokes CFD. Technical soaring. 38(1), 5-14. 2014. ISSN 0744-8996.

[11] A Selection of Experimental Test Cases for the Validation of CFD Codes, AGARD AR 303, 1994

[12] Doupník, P., Koutník, T.: Validation Task for Solution of Optimal Slotted Flap Position Using FLUENT Software, in 21th International Conference ENGINEERING MECHANICS 2015 Svratka, Czech Republic, May 11-14. : Extended Abstracts, Editors : Jiří Náprstek, Cyril Fischer. Prague : Institute of Theoretical and Applied Mechanics, Academy of Sciences of the Czech Republic, v.v.i. 2015. Paper #146, pp. 52-53. ISBN 978 80 86246-42-0, ISSN 1805-8248.