UDK: 621.6.01:519.876.5:532.517.4
COSATI: 01-01, 20-04

Comparison of the Navier–Stokes computations with the experiment for LASTA – 95 wing at high angles of attack

Mirko Kozić, PhD (Eng)[1)]

A comparison of the numerical and experimental results for LASTA – 95 wing at high angles of attack has been done. A three-dimensional Navier – Stokes code is used for computing flow field around the wing. Both unstructured and structured meshes are generated. The influence on the results of the mesh type and its resolution near the wing surface is analyzed. Comparisons pointed to the fact that reliable results can not be obtained by inadequate mesh resolution. The intention to use numerical simulation as virtual wind-tunnel does not apply in every simulation, regardless of the exact boundary, initial conditions and the solution convergence.

Key words: **fluid dynamics, Navier–Stokes equations, numerical methods, stalling incidence, aerodynamic coefficients.**

Introduction

THE numerical simulation of three-dimensional flows around wings at high angles of attack is one of the most challenging problems for all codes solving Navier-Stokes equations. This is due to very complex flow physics, characterized by early and massive separation that must be captured adequately in order to obtain reliable lift coefficient curve and drag polar. The mesh of appropriate resolution is a prerequisite of an adequate numerical simulation.

Experimental and numerical results obtained for LASTA-95 wing at high angles of attack are compared in the paper. Numerical results are obtained by using unstructured and structured meshes with different resolutions and approximately equal number of cells. Unstructured meshes composed of tetrahedrons are generated automatically, enabling adaptive meshing; however, they also require larger computer resources and the solver is much slower [1]. Structured meshes composed of prisms are much more difficult to generate, requiring therefore considerable user's skill. The difficulties increase with the complexity of geometry, [2]. However, the user can influence the mesh resolution directly. The influence of the mesh type and resolution on the numerical results, along with their correspondence with the experiment are analyzed in the paper.

Theoretical and numerical bases of the used solver

The governing equations of fluid flow represent mathematical statement of the conservation laws of physics, i.e. conservation of mass, momentum and energy. Assuming that the viscous stresses are proportional to the rates of deformation, the Navier-Stokes equations, representing the approximation of momentum conservation are obtained.

The finite volume method is used as a numerical solution technique. It consists of formal integration of the governing equations of the fluid flow over all the control volumes (cells) of the solution domain. In order to convert integral equations into a system of the algebraic equations, a discretisation is done. It involves the substitution of different approximations with the terms in the integrated equation. Solution of the system of the algebraic equations finally gives flow variables (velocity, pressure, temperature etc.) that are defined at the node inside each cell.

The code uses two different approaches in treating flow field turbulence, [3]. The first approach includes turbulence models from the simplest (algebraic equations), to the most complex (partial differential equations governing Reynolds stresses). This approach results from time averaging of the Navier-Stokes equations that yields the Reynolds averaged equations with additional six unknown quantities known as Reynolds stresses. They appear due to the momentum exchange of macroscopic fluid particles in the turbulent flow. The system of equations is not closed, and Reynolds stresses must be modelled in terms of mean flow properties that are known, to obtain a sufficient number of equations for all of the unknowns. The choice of a turbulent model mainly depends on its possibility to resolve the essence of the flow physics.

The second approach is a Large Eddy Simulation in which the large eddies are computed and the smallest eddies are modelled by space filtering. This approach is based on the assumption that the largest eddies are highly anisotropic and flow dependent due to their strong interaction with the mean flow, while the smallest eddies are isotropic and at these scales the viscous dissipation becomes important. LES is considerably time consuming compared to the first approach.

Flow regime of the numerical simulation

The computations are performed at zero yaw angle and only include one-half of the symmetric wing geometry delimited by a symmetry plane. The free-stream Mach

1) Military Technical Institute (VTI), Ratka Resanovića 1, 11132 Belgrade

number is 0.286, the Reynolds number is $8 \cdot 10^6$ based on the wing reference chord. According to the Mach and Reynolds numbers turbulent and compressible flow is modelled. Although the Mach number is below the compressibility limit, local compressibility effects can not be neglected near the leading edge where large acceleration appears. Also, the model of compressible flow includes energy equation, which accelerates solution convergence for steady flows.

Analysis of the mesh type and its resolution

Two unstructured meshes composed of $0.5 \cdot 10^6$ and $0.7 \cdot 10^6$ tetrahedrons were used in the first phase of investigation. Due to the of complex flow field at high incidences, both meshes have proven to be inadequate, regarding the drag coefficient value. The basic reason was insufficient mesh resolution near the wing surface, i.e. too large grid elements that could not resolve wall turbulence and other essential flow features, [4]

The drag force coefficient decreases with the Reynolds number increasing and relative roughness decreasing. However, unstructured meshes used in the numerical flow simulation around the smooth wing surface, gave the drag coefficient larger than experimental obtained for the whole aircraft model.

This fact shows that the used unstructured meshes are not good enough, because the essential flow features are not resolved at high incidences, so unreliable drag coefficient curve and polar are obtained.

The attempt to solve the problem by mesh adaptation did not succeed, since wall turbulence was not captured. This is a logic result of using unstructured meshes with insufficient number of elements especially near the wall. It was shown in [1] that the least number of cells of an adequate unstructured mesh is of order 10^7, i.e. well beyond the available computer resources in MTI.

That is why further investigation was carried out using a structured mesh with approximately $1.1 \cdot 10^6$ cells. This mesh has high resolution near the wing surface and the non-dimensional wall distance of the first cell row centroids is $y^+ = 0(1)$. In this way, the total number of cells is held under the control and also the essential features of the flow field are obtained.

Results

The computation was done using the following reference values of geometrical and flow quantities, while complete wing geometry is given in [5].

$$p_{ref} = 101325 \text{ N/m}^2$$

$$\rho = 1.225 \,\text{kg/m}^3$$

$$T = 288 \text{ K}$$

$$V_{ref} = 97.2 \text{ m/s}$$

$$S = 6.45 \,\text{m}^2$$

$$l_a = 1.418 \text{ m}$$

Experimental results in the form of lift and drag force coefficients as function of the angle of attack are available from VTI wind-tunnel T-35, [6]. They are defined as

$$C_z = \frac{R_z}{1/2\, \rho V_{ref}^2\, S}$$

$$C_x = \frac{R_x}{1/2\, \rho V_{ref}^2\, S}$$

In Figures 1-2 the numerical values for the wing and experimental results of the lift and drag coefficients for the whole airplane are compared as a function of the angle of attack.

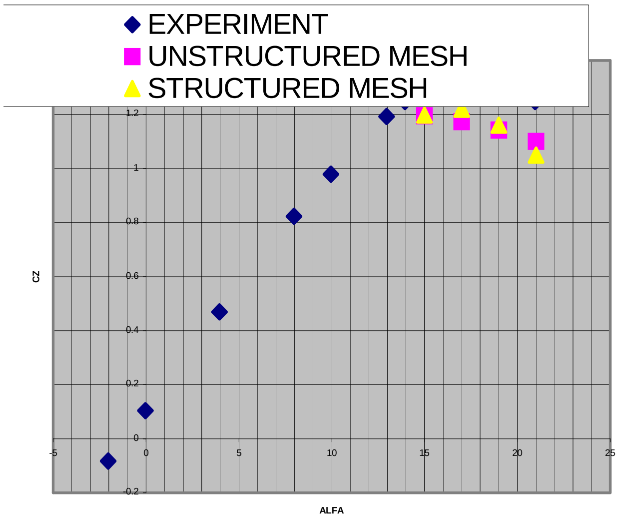

Figure 1. Comparison of experimental and computed lift coefficients as a function of the angle of attack

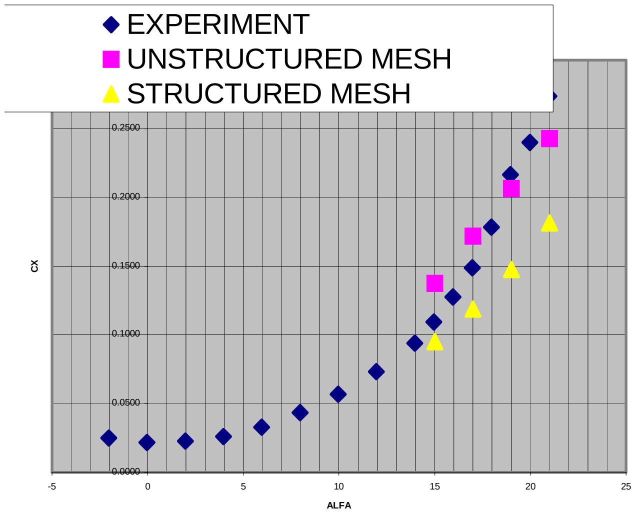

Figure 2. Comparison of the experimental and computed drag coefficients as function of the angle of attack

The flow separation from the wing surface occurs at high incidences. The position of separation line strongly influences pressure and viscous drag. Therefore, it is of utmost importance to determine the position of separation line on the wing surface as accurately as possible.

Conclusion

The paper presents a study of the influence of mesh resolution on the numerical results. The comparison of the numerical and experimental results for LASTA-95 wing have shown that reliable lift and drag coefficient at high angles of attack can not be obtained using meshes with inadequate resolution.

Regardless of the mesh size, a numerical simulation only approximates the complex flow field created over a wing. Results are more reliable with greater resolution and can reduce the quantity of wind tunnel testing considerably. The research clearly showed that high fidelity of solution can not be obtained with each simulation that satisfies the boundary and initial conditions, giving convergent solution.

Making sophisticated numerical simulations that would resolve the essence of flow is an unsolvable task without a good knowledge of flow physics, numerical methods and turbulent models.

References

[1] MAVRIPLIS,D.J., PIRZADEH,S.: *Large-scale parallel unstructured mesh computations for three-dimensional high-lift analysis*, Journal of Aircraft, 1999, Vol 36, No.6, pp.987-998.

[2] KOK,J.C., BOERSTOEL,J.W., KASSIES,A., SPEKREIJSE,S.P.: *A robust multi-block Navier-Stokes flow solver for industrial applications*, NLR TP 96323, 1996.

[3] WILCOX,D.C.: T*urbulence Modeling for CFD. 2nd ed. DCW Industries*, Inc, 1994, ISBN 0-9636051-0-0

[4] ANDERSON,W.K., BONHAUS,D.L., McGHEE,R.J., WALKER,B.S.: *Navier-Stokes computations and experimental comparisons for multielement airfoil configurations*, Journal of Aircraft, 1995, Vol 32, No.6, pp.1246-1253.

[5] MOLOVIC,V., MARJANOVIĆ,A., ZDRAVKOVIĆ,M., VELIMIROVIĆ,K.: *Calculation of aerodynamic characteristics and flight performance of the airplane LASTA-95*, V3-0464-P-023, MTI, 2002.

[6] OCOKOLJIĆ,G., ILIĆ,B.: *Airplane LASTA-95 model 1:5 testing, in windtunnel T-35*, VTI VSCG, 2005.

Received: 28.01.2006.

Poređenje rezultata Navije-Stokovog solvera sa eksperimentalnim za krilo aviona LASTA-95 pri velikim napadnim uglovima

Izvršeno je poređenje i analiza eksperimentalnih i numeričkih rezultata dobijenih za krilo aviona LASTA-95 pri velikim napadnim uglovima. Za izračunavanje strujnog polja oko krila korišćen je softver za rešavanje Navije-Stoksovih jednačina za trodimenzijsko strujanje. Generisane su kako strukturisane tako i nestrukturisane mreže. Analiziran je uticaj tipa mreže i njene rezolucije uz površinu krila, na dobijene rezultate. Poređenja su pokazala da se pouzdani rezultati ne mogu dobiti mrežama sa neadekvatnom rezolucijom. Težnju da numerička simulacija posluži kao virtuelni aerotunel, ne ispunjava svaka numerička simulacija, bez obzira na tačne granične i početne uslove kao i konvergenciju rezultata.

Ključne reči: **dinamika fluida, Navije-Stoksove jednačine, numeričke metode, kritični napadni ugao, aerodinamički koeficijenti.**

Сравнивание результатов вычисления Навиер-Стокса с экспериментальным для крыла самолёта "Ласточка-95" при больших углах атаки

В настоящей работе проведено сравнение и анализ экспериментальных и численных результатов, полученых для крыла самолёта "Ласточка-95" при больших углах атаки. Для вычисления поля потока около крыла пользовано программное обеспечение для решения уравнений Навиер-Стокса для трёхразмерных потоков. Генерированы и структурные и неструктурные сетки. Тоже проведён анализ влияния типа сеток и их резолюции при поверхности крыла, на полученные результаты. Сравнения доказали, что невозможно получить достоверных результатов с употреблением сеток с несоответствующей резолюцией. Интенцию, чтобы численное моделирование было возможно употребить в роли виртуальной аэродинамической трубы, не выполняет каждое численное моделирование, не смотря на точные пограничные и исходные условия, а также и на сходимость результатов.

Ключевые слова: динамика жидкостей, уравнения Навиер-Стокса, численные методы, критически угол атаки, аэродинамические коэффициенты.

Comparaison des résultats de Navier-Stokes computations avec les essais pour l'aile de l'avion Lasta-95 aux grandes angles d'attaque

Ce papier traite l'analyse et la comparaison des résultats expérimentaux et numériques obtenus pour l'aile de l'avion LASTA-95 aux grandes angles d'attaque. Pour calculer le champ du courant autour de l'aile on a utilisé le logiciel servant à résoudre les équations de Navier-Stokes pour le courant à trois dimensions. On a produit les réseaux structuraux et non-structuraux. On a également analysé l'influence du type de réseau et de ses résolutions près de la surface de l'aile sur les résultats obtenus. Les comparaisons ont démontré que les résultats sûrs ne peuvent pas être obtenus par les réseaux à résolution non-adéquate. L'intention que la simulation numérique serve de soufflerie virtuelle ne réalise pas chaque simulation numérique malgré la précision des conditions limites initiales et en dépit de la convergence des résultats.

Mots clés: **dynamique des fluides, équations de Navier-Stokes, méthodes numériques, angle d'attaque critique, coefficients aérodynamiques.**