International Science and Technology Journal
المجلة الدولية للعلوم والتقنية

Volume 32 العدد
Part 2 المجلد
July 2023 يوليو

المجلة الدولية للعلوم والتقنية
International Science and Technology Journal
ISTJ

تم استلام الورقة بتاريخ: 15/ 6 /2023م وتم نشرها على الموقع بتاريخ: 20/ 7 /2023م

Investigation of Turbulence Models for Flow Simulation Around A multi-element Airfoil NACA23012 at Varying Angles of Attack

Asya A. Gabbasa
Faculty of Oil and Gas Engineering, University of Zawia
a.gabbasa@zu.edu.ly

ABSTRACT

The study of flow development has been carried out since the last century. It is very important in fluid engineering because of its numerous applications. The flow development depends on the geometry and dynamical parameters. In the present paper, an approach has been made for dealing with turbulent flows within multi-element airfoil. The[P1] turbulent flow around a multi-element airfoil using the wall y+ As a mentoring for a suitable grid configuration and corresponding turbulence models are investigated by using Fluent. This work was done under some assumption of constant viscosity, incompressible flow, and low Reynolds numbers. Spalart-Allmaras (SA), the standard k-ε, Reynolds stress model (RSM), and renormalization group (RNG) k-ε turbulence models are used to solve the closure problem . Their behaviors collectively with the accompanying near-wall

treatments at different angles of attack are investigated for wall y+≈1 overlaying the viscous sublayer and Y+< 4 to 5 in the buffer region. The results show the maximum value of Y+ at these attack angles is almost Y+<5. This shows that the Fluent results were consistent. Therefore, the resolution of the near-wall grid is acceptable.

Keywords: Turbulence Model, Computational Fluid Dynamic, Airfoil, Wall Y+≈1, NACA23012.

International Science and Technology Journal
المجلة الدولية للعلوم والتقنية

Volume 32 العدد
Part 2 المجلد
July 2023 يوليو

المجلة الدولية للعلوم والتقنية
International Science and Technology Journal
ISTJ

تم استلام الورقة بتاريخ: 15/ 6 /2023م وتم نشرها على الموقع بتاريخ: 20/ 7 /2023م

التحقيق في نماذج الاضطراب لمحاكاة التدفق حول الجنيح متعدد العناصر في زوايا مختلفة من الهجوم NACA**23012**

د. اسيا على قباصه

كلية هندسة النفط والغاز , جامعة الزاوية

a.gabbasa@zu.edu.ly

الملخص

تم إجراء دراسة تطوير التدفق منذ القرن الماضي. وهو مهم جدا في هندسة الموائع بسبب تطبيقاته العديدة. ويعتمد تطوير التدفق على شكل النموذج والمعاملات الديناميكية. في هذه الورقة ، تم اتباع نهج للتعامل مع التدفقات المضطربة داخل الجنيح متعدد العناصر. ويتم فحص التدفق المضطرب حول الجنيح متعدد العناصر باستخدام الجدار **y** + كتوجيه لتكوين شبكة مناسب ونماذج الاضطراب المقابلة باستخدام **Fluent**. تم تنفيذ هذا العمل تحت بعض الافتراض باللزوجة الثابتة والتدفق غير القابل للضغط وأرقام رينولدز المنخفضة. يتم استخدام نماذج الاضطراب **(SA K–Allmaras)** ، ونموذج **k–ε** القياسي ، ونموذج رينولدز للإجهاد **(RSM)** ، ومجموعة إعادة التطبيع **(RNG) k–ε** لحل مشكلة الإغلاق.

تم التحقيق في سلوكياتهم بشكل جماعي مع المعالجات القريبة من الجدار المصاحبة في زوايا مختلفة من الهجوم للجدار ≈ + 1**y** الذي يغطي الطبقة الفرعية اللزجة و **Y** + > 4 إلى 5 في المنطقة العازلة. وتظهر النتائج أن القيمة القصوى ل **Y** + عند زوايا الهجوم هذه هي تقريبا **Y** + <5. هذا يدل على أن نتائج **Fluent** كانت مناسبة جدا. لذلك ، فإن دقة الشبكة القريبة من الجدار مقبولة.

Introduction

Computationally expensive simulation codes based on mathematical models of a relevant system are in widespread use

تم استلام الورقة بتاريخ: 15/ 6 /2023م وتم نشرها على الموقع بتاريخ: 20/ 7 /2023م

throughout the engineering industry. For instance, in the field of computational fluid dynamics (CFD), a single evaluation of a model may take several hours of computer run time. With the help of modern computer facilities, numerous complicated and mathematically intensive problems are now readily solved (Gabbasa et al, 2013).

Most of the time spent on a computational fluid dynamics (CFD) project is usually devoted to successfully creating, a mesh for the domain. Allowing a compromise between the required accuracy and the cost of the solution. This time-consuming procedure is considered a bottleneck in the analysis.

To determine the most accurate mesh, it is preferable to perform operating tests on different mesh sizes and configurations and match the numerical solution as closely as possible to the experimental data, in the so-called grid-independent test.

The presence of the walls strongly affects turbulent flows, where viscosity-affected areas have large gradients in solution variables and the exact width of these areas determines the successful prediction of finite flows by the wall. (Gerasimov, 2006).

Best practice recommendations for mesh networks based on calculated wall Y+ have been successfully developed for cases where reliable empirical data may not be available for validation. (Salim et al, 2009).

Their study was conducted for a two-dimensional (2D) problem covering both un turbulent and turbulent flows on a solid edge bounded by a flat smooth wall at Re=17,000 based on average flow velocity and ridge height[P1] , and their recommendations include suggested behavior and the use of Navier-Stokes (RANS) models and near-wall treatments using Fluent .

To complicate 3D turbulence, turbulent flow around a wall-mounted cube has been studied experimentally, for an advanced turbulent channel flow with low Reynolds numbers of $2750 < Re < 4970$. (Meiders et al,1999). The case was chosen due to its simple

تم استلام الورقة بتاريخ: 15/ 6 /2023م وتم نشرها على الموقع بتاريخ: 20/ 7 /2023م

geometry but complex flow structures and represents a general engineering configuration that is relevant to many engineering applications ranging from prediction of wind loading on structures to cooling of turbines and electronic components in circuit boards. Similarly, a significant number of numerical studies using simple RANS models such as k-ε, unsteady RANS, large eddy simulation (LES), and the more complicated Direct Numerical Analysis (DNS) have been performed for such a flow configuration that has been widely used for benchmarking purposes to validate turbulent models and numerical methods. Recently; heat transfer for the same flow condition has been included. (Ratnam and Vengadesan 2008). In all the numerical analyses, particularly those using RANS formulation, the accuracy of the results depended on the turbulent models used, the near-wall treatments applied, the discrimination schemes used, and the convergence criteria set, among other solver factors. The selection of the best meshes, accompanying turbulence models, and near-wall treatments with Fluent are particularly useful as a guide for situations where empirical verification or initial design considerations may not be available and where alternative experimentation is expensive.
The present work aims to investigate study from the turbulent flows over a multi-element airfoil using the wall y+ as guidance for suitable grid configuration and corresponding turbulence models at varying angles of attack are investigated using Fluent. This increases confidence in using CFD trading packages for industrial simulations rather than relying solely on experimentation when dealing with wall-bounded complex turbulent flows.

Near-Wall Treatment

Areas close to the wall have more gradients in solution variables, momentum, and other numerical transfers occur more strongly. From Figure 1, it can be seen that the area affected by viscosity (the inner layer in this situation) includes three zones with corresponding walls (White, 2006) particularly the:

تم استلام الورقة بتاريخ: 15/ 6 /2023م وتم نشرها على الموقع بتاريخ: 20/ 7 /2023م

- Viscous sub layer (y+<5)
- Buffer layer or blending area (5< y+<30)
- Fully turbulent or log-law region (y+>30 to 60)

The wall y+ is a dimensionless distance much like the local Reynolds number, frequently used in CFD to explain how rough or accurate the grid is for a given flow. It is the ratio of turbulent effects and laminar effects within the cell. Too close to the wall, viscous damping reduces transverse velocity fluctuations, at the same time as kinetic blocking reduces natural fluctuations. However, towards the outside of the region close to the wall, the disturbance rapidly will increase.

Accurate presentation of the flow in the close-to-wall area determines the successful prediction of wall-bounded turbulent flows. Values of y+ close to the lower sure (y+≈30) are most perfect for wall capabilities whereas y+≈1 is most acceptable for close-to-wall modeling.

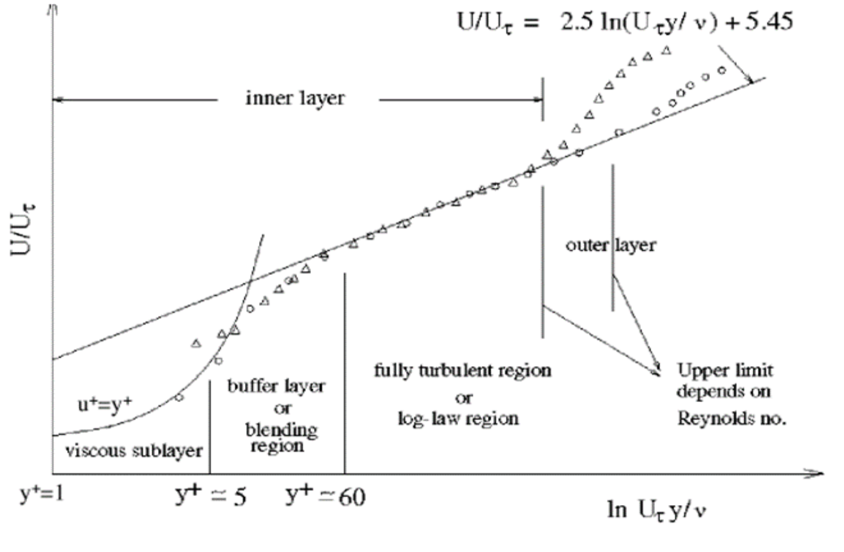

Figure 1. Subdivisions of the near-wall region. [Frank White,2006].

International Science and Technology Journal
المجلة الدولية للعلوم والتقنية
Volume 32 العدد
Part 2 المجلد
July 2023 يوليو
تم استلام الورقة بتاريخ: 15/ 6 /2023م   وتم نشرها على الموقع بتاريخ: 20/ 7 /2023م

Numerical Simulations:

1-Model Generation

This section examines the systematic methodology used forthe creation of models and analysis. The first step was to construct the geometry of the airfoil and the external airfoil flap, then to create meshes for the geometry of the foil. This was done using the Gambit software package[P4] . This package is used for pre-processing models used for computational fluid dynamics (CFD) to generate the geometry and meshes that are then imported into another program. The information created in Gambit is imported into Fluent and used to solve the governing fluid dynamic equations for the model.

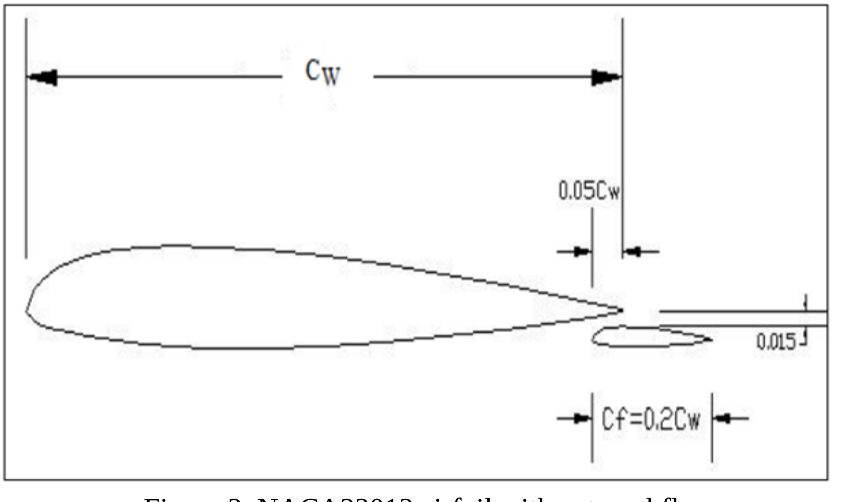

Figure 2. NACA23012 airfoil with external flap

2- Mesh Generation

Once the initial geometry for the airfoil has been created, the next stage is the generation of the mesh. A mesh (or grid) refers to how a domain is split up into sub-domains, known individually as

تم استلام الورقة بتاريخ: 15/ 6 /2023م وتم نشرها على الموقع بتاريخ: 20/ 7 /2023م

elements or cells[P5] . That means the sub-domain is the unit area of mesh elements. When generating a mesh, the entire mesh area is the Domain and the mesh elements that partition the mesh are the Sub-Domains. The creation of a mesh is essential for describingthe flow of fluid in a domain through an analytical solution. Once individual cells are created, equations can then be solved for the cells.

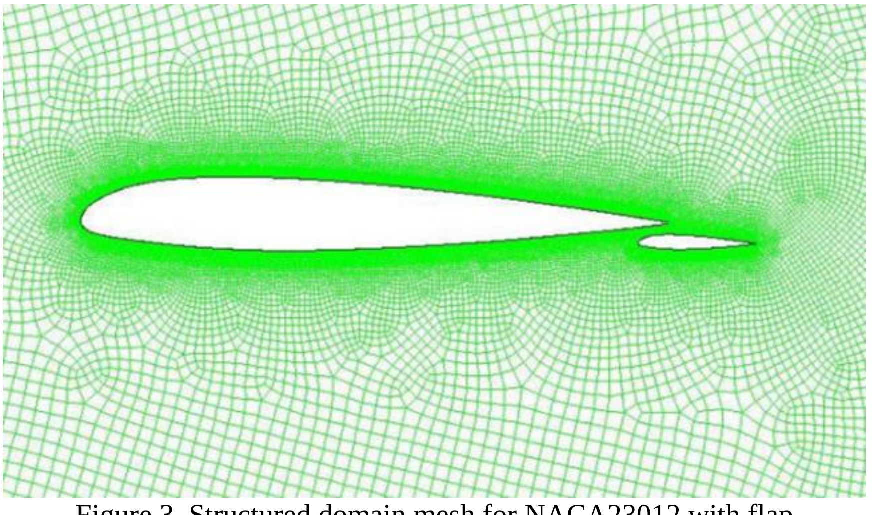

Figure 3. Structured domain mesh for NACA23012 with flap

Table 1: Mesh entities for a two-dimensional model

Total mesh cells	**172,691**
Total mesh faces	**347,113**
Total mesh nodes	**174,421**

3- Solver

For this article, a two-equation turbulent model was used to solve the governing equations. Here, the Navier-Stokes equations were used in their integral form, with the assumption of constant viscosity and incompressible flow. The realizable k-ε model for a single

تم استلام الورقة بتاريخ: 15/ 6 /2023م وتم نشرها على الموقع بتاريخ: 20/ 7 /2023م

transport equation was used. This allowed for the individual determination of the turbulent kinetic energy and the rate at which it dissipated. This model was particularly relevant for this study, as it made use of enhanced wall treatment.
The incompressible steady-state two-dimensional flow was assumed for this work which allowed for variables to be solved using a Semi-Implicit Method for Pressure-Linked Equation as the algorithm. The interpolating scheme used in this study was a first-order upwind method.

Results

The presence of walls affects turbulent flows. The k-εturbulence model is valid when away from the wall. Therefore, there is a need for special treatment in order to ensure that this model is valid when next to walls. The current work used a specific treatment, which was enhanced wall treatment. This method refers to a near-wall modeling method, which joins two-layer models (buffer layer and sub-layer). Y^+ values depend on the grid resolution, as well as the flow Reynolds number, and their definition is valid in the wall-adjacent cells. This in turn determines how the calculation of the shear stress is performed. In a laminar sublayer of enhanced wall treatment, Y+ values should be Y+=1. Nevertheless, higher Y+ values are acceptable provided they are within the viscous sublayer (Y+<4 to 5). Figures 4 to 6 are illustrated Different values of y+ at varying attack angles.
The Figures show Y+ values in the bottom and top airfoil and flap surfaces. The maximum value of Y+ at these attack angles is almost similar the values are Y+<5. This shows that the Fluent results were consistent. Therefore, the resolution of the near-wall grid is acceptable.

تم استلام الورقة بتاريخ: 15/ 6 /2023م وتم نشرها على الموقع بتاريخ: 20/ 7 /2023م

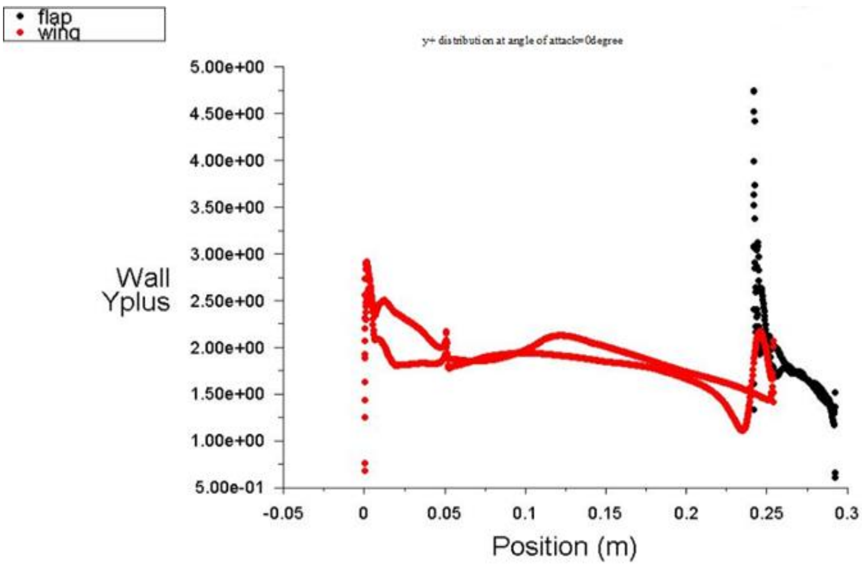

Figure 4. Y+ distribution at angle of attack =0 degree

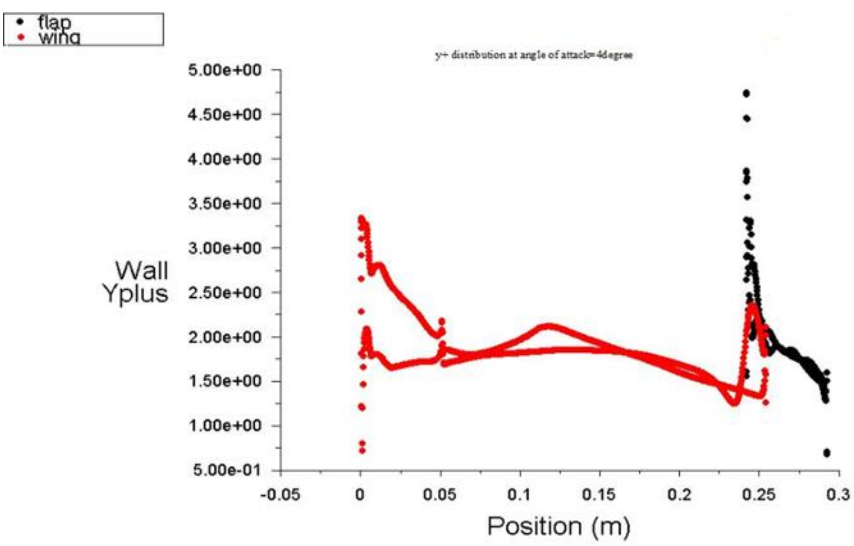

Figure 5. Y+ distribution at angle of attack = 4 degrees

تم استلام الورقة بتاريخ: 15/ 6 /2023م وتم نشرها على الموقع بتاريخ: 20/ 7 /2023م

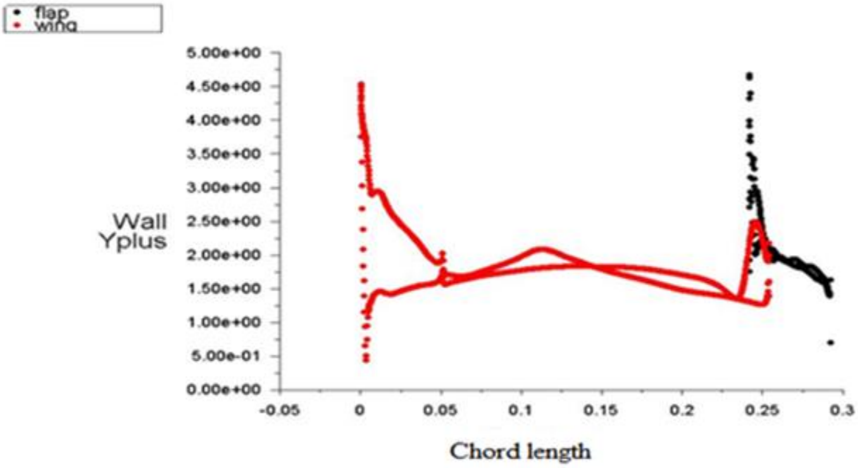

Figure 6. Y+ distribution at angle of attack = 8 degrees

Conclusion

The article has concluded that the non-dimensional wall y+ is the right desired criterion for figuring out an appropriate mesh configuration and turbulence model, coupled with near-wall treatment, that ends in accurate computational predictions in fluent. A mesh size that resolves the log-law region is sufficient for computation, without incurring added time or attempt by means of refining into the viscous sublayer. It's also definitely useful to avoid resolving the buffer (mixing)region as neither wall functions nor near-wall modeling debts for it accurately.

References

A.V. Gerasimov,2006, Modeling Turbulent Flows with Fluent,Europe, ANSYS, lnc.

Frank M. White,2006, Viscous Fluid Flow,Mc Graw Hill, Third Edition.

تم استلام الورقة بتاريخ: 15/ 6 /2023م وتم نشرها على الموقع بتاريخ: 20/ 7 /2023م

Gabbasa A., Jawad,B., Liu, L., and Arslan, S.,2013, Numerical Study of the Aerodynamic Characteristics of a Multi-Element Airfoil NACA23012, SAE Int.J., Aerosp,6(1) , doi, 10.4271/2013-01-1410.

Meinders, E.R, Hanjalic, K., and Martinuzzi, R.,1999, Experimental study of the local convective heat transfer from a wall-mounte cube in a turbulent channel flow, Trans ASME J. Heat Transfer 121, 564-573.

Ratnam, G.S., and Vengadesan, S.,2008, Performance of two-equation turbulence models for prediction of flow and heat transfer over a wall-mounted cube, Int. J. Heat and Mass Transfer, 51, 2834-2846.

Salim M. Salim, and S.C. Cheah, 2009, Wall y+ Strategy for Dealing with Wall-bounded Turbulent Flows, Proceedings of the International Multi-Conference of Engineers and ComputerScientists, Vol II. IMECS 2009, March 18 - 20, Hong Kong.