©2022 Published in 10th International Symposium on Innovative Technologies in Engineering and Science 21-23 October 2022 (ISITES2022 Bursa - Turkey)
https://doi.org/10.33793/acperpro.05.03.3850

CFD Strategies for Transonic Flows with OpenFOAM Using High Skewed Tetrahedral Meshes

*1,2Halil Bulus and 2Sertac Cadirci
1ASELSAN A.S., Ankara, Turkey
2Department of Mechanical Engineering, Istanbul Technical University, 34437, Istanbul, Turkey

Abstract:

The usage of OpenFOAM flow solver program can be challenging for some industrial applications as it is highly sensitive to the quality of the solution mesh. Therefore, users are inherently encouraged to use low skewed meshes which have six or more faces. The implementation of hexahedral elements helps to accurately model a physical problem and speed up the convergence by allowing the solution meshes to be aligned orthogonally along a curved geometry. However, tetrahedral elements are mostly the first choice in order to easily mesh complex and curved structures encountered in industrial problems, since they do not cause any distortions. However, in some cases, the implementation of highly skewed tetrahedral elements may be a necessity and thus a challenging issue for OpenFOAM users. Although the solution approaches can be varying due to the nature and necessities of the flow problems, a generalized solution strategy has been put forward as the starting way for compressible, turbulent, external flow problems, and then it is aimed to reach the correct results with required minor modifications throughout the solution. This suggested solution strategy has been applied to some basic CFD benchmark cases created with highly skewed tetrahedral meshes and they have been successfully validated with experimental data. As a result, it is shown that the sensitivity of OpenFOAM to highly-skewed tetrahedral meshes can be managed with properly chosen solution strategies if external compressible flows are supposed to be solved numerically.

Key words: CFD, OpenFOAM, compressible flows, aerodynamics, high skewed meshes

1.Introduction

OpenFOAM is a C++ based, open source Computational Fluid Dynamics (CFD) program [1]. Due to its advantages and open source code features, it has been increasingly used by many researchers since it was first published in 2004 and continues to be developed by volunteers worldwide. Unlike commercial applications, OpenFOAM does not have any GUI interface and uses the Paraview application, which is also open source post-processing tool.

OpenFOAM has a large library of solvers which are specialized for a large kind of specific heat and fluid flow problems. Although the parameters and physical properties required for the problem of interest are defined within these solvers, the information needed to define the problem and the solution/solver methods are defined in specific folders. The general file structure of these folders for a representative case is shown in Figure 1.

*Corresponding author: Address: Microelectronic, Guidance and Electro-optic Sector Presidency, Aselsan A.Ş, Ankara, Turkey. E-mail address: hbulus@aselsan.com.tr

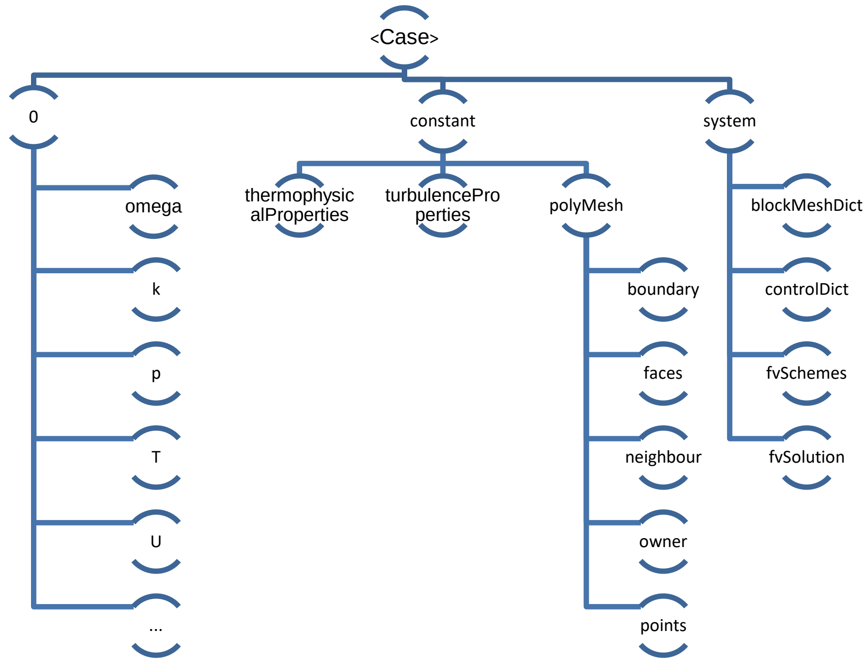

Figure 1. File structure of a representative case

2.Method

In order to converge the simulations with highly skewed tetrahedral meshes, i.e. meshes with skewness greater than 70 degrees by using second order discrete equations, a specific solution strategy has been developed. Modifications may be needed depending on the nature of the problem. In this study, the schemes and solution techniques mentioned are explained for external viscous and compressible flows, hence it is not applicable to all types of fluid flow problems.

Step 1. Gradient and divergence schemes are selected as first order. Linear solvers settings are selected for numerically stabile, but less accurate solution and high relaxation factors are implemented. Selected gradient scheme can be summarized as:

All other variables (e, Ekp, U, k, omega, nuTilda) are limited during discretization except the pressure gradient. The so-called "cellLimited" scheme guarantees that the face values will not go beyond the cell values limits during the interpolation to the faces of the solution cells. While the velocity and turbulence terms are fully limited, the energy terms are partly limited. Since pressure is not a convective term, it is not limited to add numerical diffusion to the solution. It has been observed that in some cases when the pressure term is limited, it will result in solutions not in agreement with the benchmark case. Selected divergence scheme can be summarized as:

In the first step, first order divergence schemes are selected. Although first-order discretization schemes are more stable, the correct solution may not be achieved due to the effects of high diffusivity. The so-called "Bounded" term is applied to scalar variable fields since limitation of cell values are more important than high accuracy. For the vector fields such as the velocity field, the focus is on high accuracy, so velocity term remains unbounded. The term 'Bounded' directly helps to decrease numerical oscillations during the solution. Linear solver settings can be summarized as: In the “fvSolution” file, the solution algorithms used in solver “rhoSimpleFoam” are defined. The term “nNonOrthogonalCorrectors” refers to the number of inner loop iterations of the pressure equation. It is suggested that for a low-quality mesh, this number should be higher than three.

The updated velocity by activating the term “momentumPredictor” is included in the solution strategy and method H is implemented [2], which is proposed for high Reynolds numbers. Activation of this term for incompressible low Mach number flows is not necessary since the density fluctuations are negligible.

Step 2. In this step, linear solver settings of the pressure variable are improved while the degrees of gradient and divergence schemes remainunchanged. The relaxation factors are lowered to a narrower interval and the solver algorithm is detailed. The number of “nNonOrthogonalCorrectors” is increased. By activating the term "consistent", the solution algorithm will turn to SIMPLEC. In this case, only velocity and other transport equations need relaxation coefficients. The off-diagonal velocities in the linear solution matrix are included in the equation. By activating the term “transonic”, transonic effects for compressible flows are included in the solution by adding the density correction term to the equation. [3].

Step 3. After a convergent solution has been achieved, the gradient and divergence schemes are increased to second order. In “gradSchemes” file, energy terms (h, e, Ekp) are limited by a differentiable gradient limit, “cellLimited<cubic>” term. In this way, the stability condition is provided as the residuals converge [4]. In “divSchemes” file, variables are discretized with the “limitedLinear” scheme which is a second-order discretization scheme. If it is considered in “Gaussian limitedLinear k” format, the correction formula will be $\varphi(r) = 2r/k$. The value of 'φ' ranges from 0 to 1. If k=0, φ will become 1 and the scheme will be of higher order. In this case, central difference scheme will be used. If k=1, then $\varphi(r) = 2r$ which means that the bounded diagram is in the TVD region as mentioned by Sweby; thus, the scheme will guarantee TVD [2, p. 98].

3.Results

3.1. Onera M6 Wing

In this study, the first benchmark case is taken from the 3D Onera M6 wing experiments for transonic flow which were carried out in 1979 [5]. This experiment is the one of the frequently used validation cases for CFD analysis. The freestream flow has a Mach number of 0.84 with an angle of attack of 3.06°. The surface mesh was created with commercial program 'Pointwise', and the volume meshes were created with commercial program, 'Ansys Fluent Meshing'. Wall y^+ value of the mesh was approximately 85 and nearly 2 million boundary layer elements were created. In the entire domain, totally 4 million elements were used. In order to precisely capture the gradient changes on the wing, 12 inflation layers in the boundary layer region were preferred. In Figure 2, the created mesh **is** shown across a section taken from the wing.

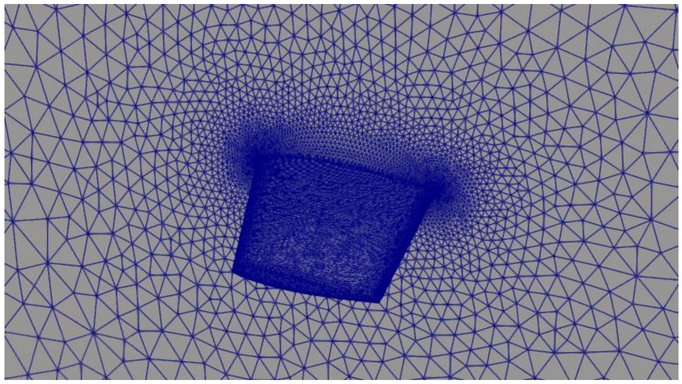

Figure 2. Mesh along the Onera M6 wing section.

It is observed that the non-orthogonality of the skewed cell surfaces is up to 77.8 degrees. The recommended mesh skewness is up to 70 degrees. In this case study, k-ω SST turbulence model was used in the flow solver 'rhoSimpleFoam'. If this solution strategy was applied, the solution has converged, and numerically obtained surface pressure coefficients (C_p) were found in good agreement with the experimental data in [5], pointing out the accuracy of the analysis results. Figure 3 shows the variation of the residuals and axial force versus iterations. As Fig.3 clearly indicates, the residuals' convergence at different levels of the solution strategy can be followed. It has been seen in the force graph that the results of the second-order numerical schemes are different from those of the first-order schemes. This indicates why second order scheme should be preferred to get accurate results for high Reynolds number viscous flows.

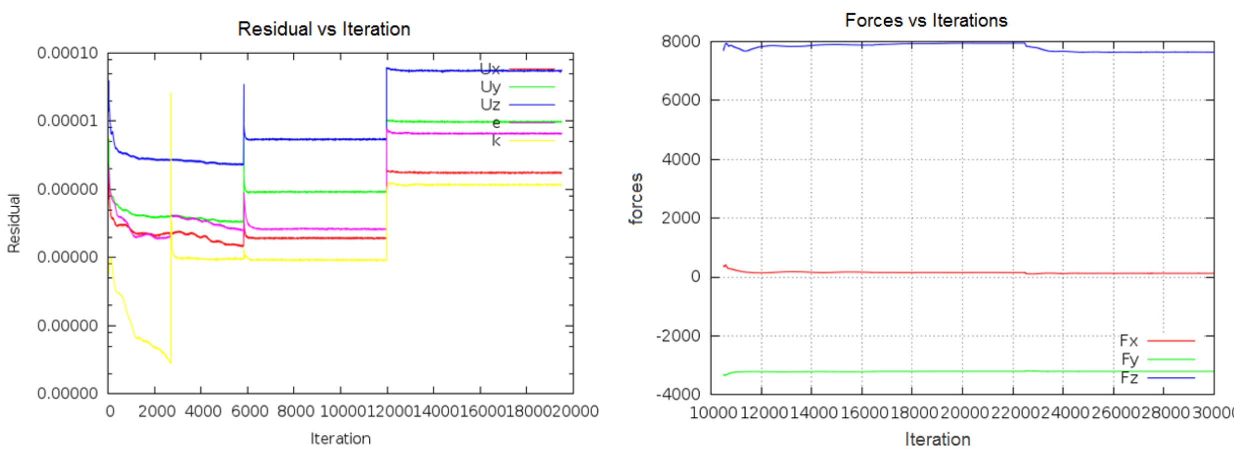

Figure 3. Residual graph (left), Forces graph (right).

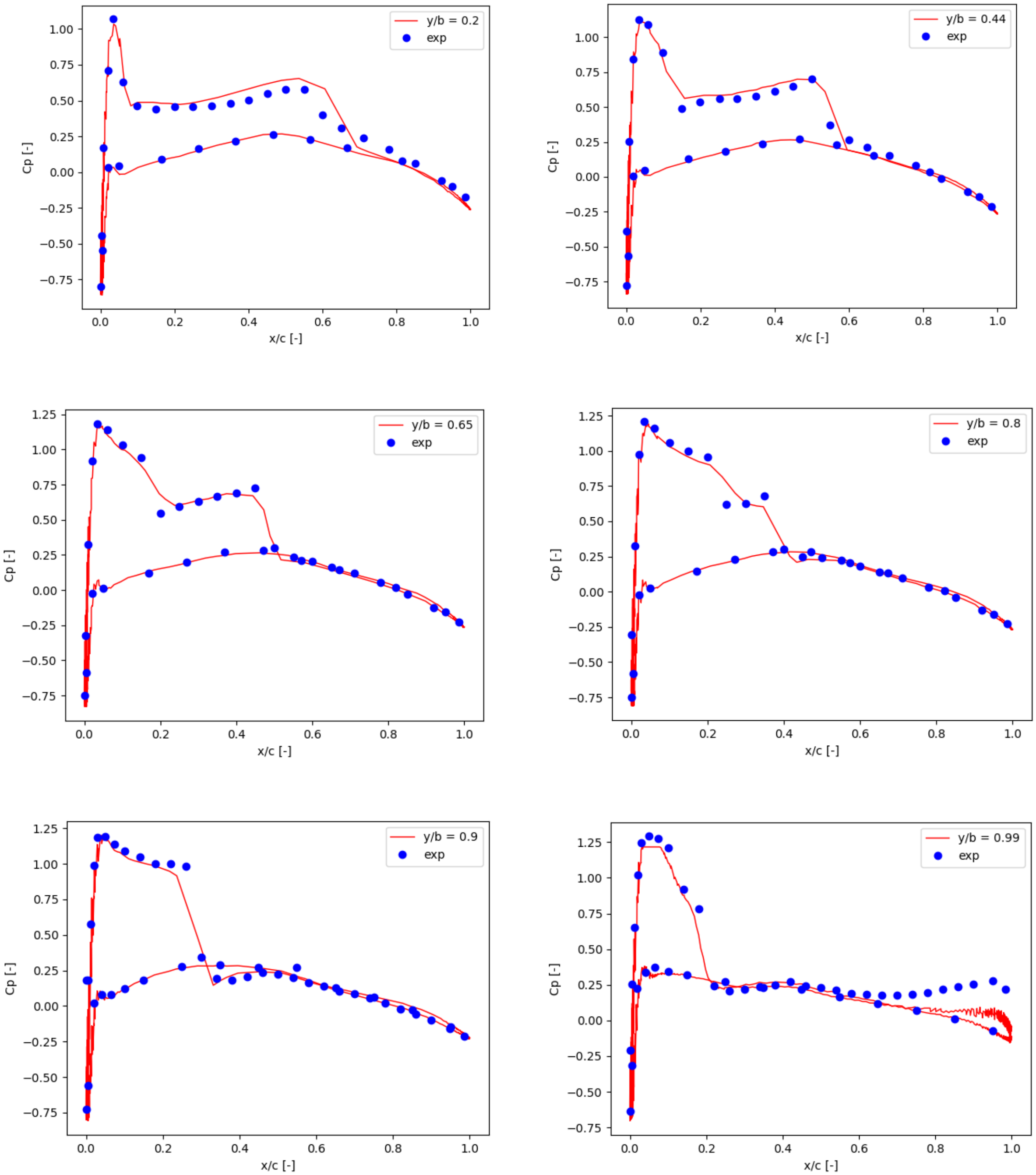

Figure 4. C_p distribution over the Onera M6 wing at various the spanwise sections

Figure 4 shows the C_p distributions at various spanwise sections of the wing which overlap with the experimental data. Only at the trailing edge (y/b=0.99) there are some deviations between the CFD results and experimental data which is associated with the mesh-quality. It can be predicted that these differences can be decreased when the meshes are improved at the tip.

3.2. DLR-F6 Aircraft

The second benchmark case consists of the CFD simulations of the transonic flow over 3D DLR-F6 aircraft where the experiments were performed in 2006 [6]. The freestream flow has a Mach number of 0.75 Mach and the angle of attack of the flow is 0.25°. The mesh-non-orthogonality test showed highest skewness of the mesh elements up to 71.8 degrees. In the second case study, Spallart Allmaras turbulence model was used in the flow solver 'rhoSimpleFoam'.

Similar to the first case study, the surface mesh was created with commercial program 'Pointwise', and the volume meshes were created with commercial program, 'Ansys Fluent Meshing'. Wall y^+ value of the mesh was kept around 21. In the boundary layer region, 1.7 million elements were implemented, thus the entire computational domain consisted of nearly 3.2 million elements. In order to correctly estimate the gradient changes on the wing section, 12 inflation layers in the boundary layer region were created. In Figure 5, the created mesh on the surface and in the fluid domain is shown.

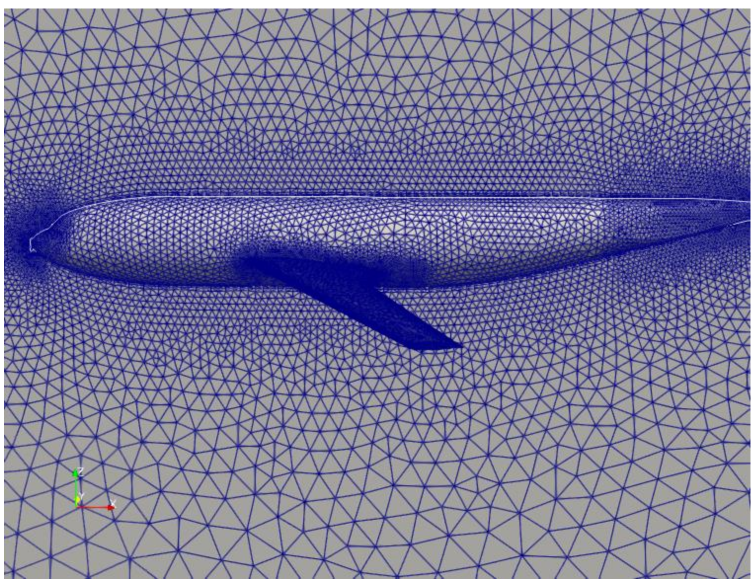

Figure 5. Mesh of the DLR-F6 wing section.

Formerly explained solution strategy has been adopted in the second case study to achieve converged solutions. The CFD results have been evaluated again in terms of surface pressure coefficient distributions on various spanwise sections of the wing and as Figure 6 demonstrates, they showed satisfactory agreement with the measurements taken from the tests in [6].

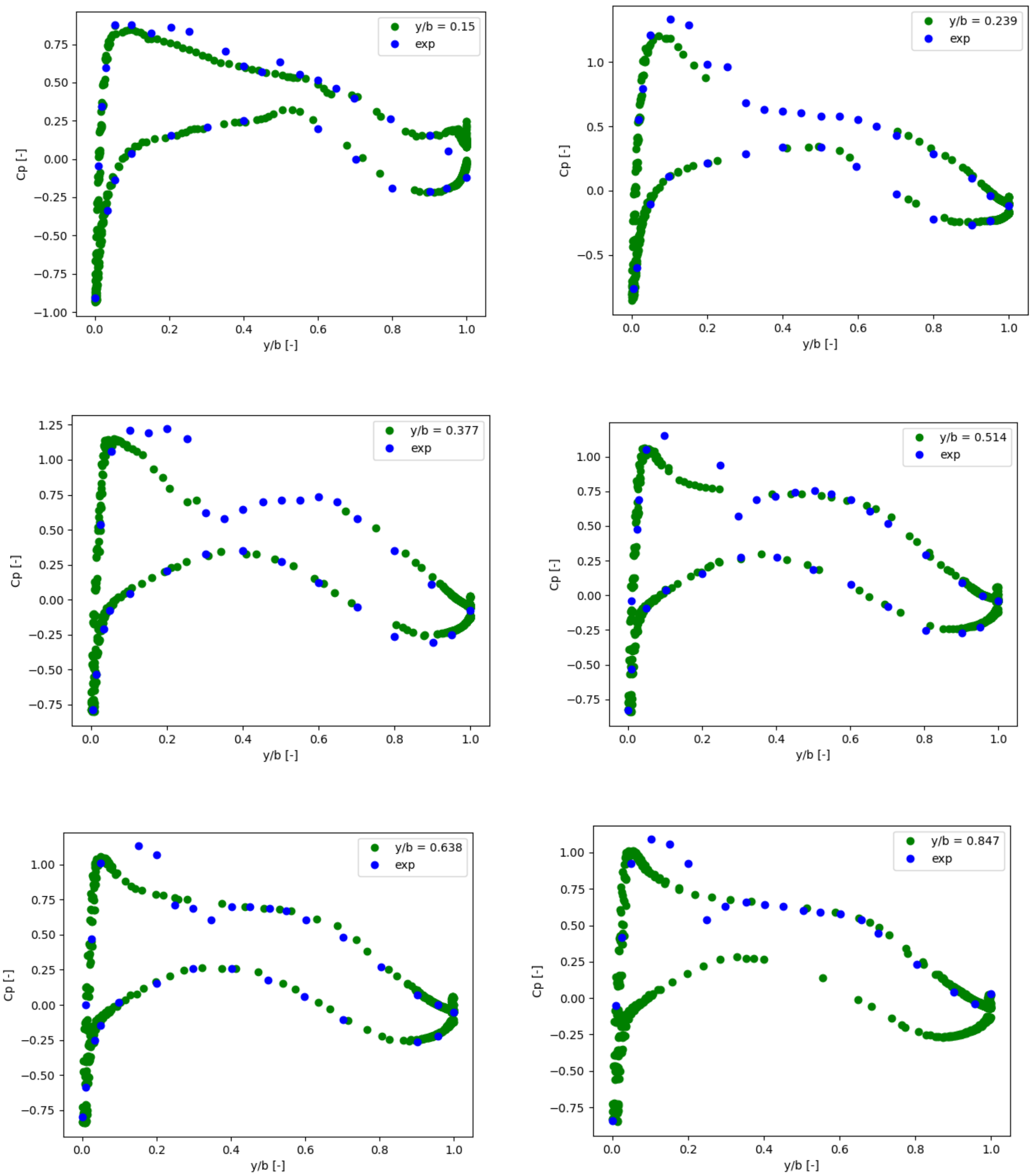

Figure 6. C_p distribution over the DLR-F6 wing at various the spanwise sections

4. Discussion

This study aims to show that using OpenFOAM with tetrahedral meshes with high skewness can cause problems in convergence and lead to incorrect solutions. On the contrary to commercial CFD programs, OpenFOAM does not allow easy access to the solution of problems in which high Reynolds number flows should be solved with highly skewed tetrahedral elements. Thus, a helpful solution strategy has been put forward for some specific case studies and it is shown that OpenFOAM can give correct results even for high skewed tetrahedral meshes. The benefits of the proposed solution strategy have been demonstrated for turbulent, compressible, transonic flows for Onera M6 wing and DLR-F6, which are the two main CFD validation cases in the literature. The results of the analyzes showed C_p distributions mostly overlapping with the experimental data, emphasizing that high mesh sensitivity of OpenFOAM can be managed with properly chosen solution levels and tetrahedral meshes can be used for a very wide range of problems with different complexities.

References

[1] User guide, OpenFOAM, https://www.openfoam.com/documentation/user-guide (accessed February 28,2022).

[2] Jasak, H., Error estimation in the finite volume method with application to fluid flows PhD Thesis, Imperial College London, 1996.

[3] RhoSimpleFoam-OpenFOAMWiki, rhoSimpleFoam, https://openfoamwiki.net/index.php/RhoSimpleFoam (accessed February 28,2022).

[4] User guide, Ansys Fluent, https://www.afs.enea.it/project/neptunius/docs/fluent (accessed February 28,2022).

[5] Schmitt, V. and Charpin, F., Pressure Distributions on the ONERA-M6-Wing at Transonic Mach Numbers, Experimental Data Base for Computer Program Assessment. Report of the Fluid Dynamics Panel Working Group 04, AGARD AR-138, May 1979.

[6] Gatlin, Greg & Rivers, Melissa & Goodliff, Scott, & Rudnik, et al., Experimental Investigation of the DLR-F6 Transport Configuration in the National Transonic Facility, 2008,10.2514/6.2008-6917.