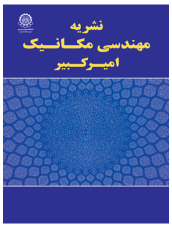

Amirkabir Journal of Mechanical Engineering

Amirkabir J. Mech. Eng., 54(5) (2022) 217-220
DOI: 10.22060/mej.2022.20472.7237

Simulation of Steady Incompressible Flow around a NACA0015 Airfoil Using Actuator Surface Method and Mass Corrected Interpolation Technique

H. Ettehadi, M. Tebyan , H. Alisadeghi*

Department of Aerospace Engineering, K.N. Toosi University of Technology, Tehran, Iran.

ABSTRACT: In recent years, actuator methods in aerodynamic simulations have been favored by researchers. These methods can significantly reduce the computational effort compared to full-scale body resolving simulations. They are also more accurate than conventional methods that use simplified models. In this study, an actuator surface model is used to simulate flow around an airfoil in a steady two-dimensional incompressible flow. In these models, the geometry of the airfoil is represented by volume forces distributed along the airfoil chord. For this purpose, the collocated method of mass corrected interpolation method is coupled with the Actuator Surface Model. To determine the accuracy of the results, the actuator surface method is compared with the full- computational fluid dynamics simulation method. Besides, a new study is presented to investigate the effect of changing different parameters of the actuator surface model on the accuracy of results. Finally, pressure and vorticity contours are plotted, and obtained results are compared with full- computational fluid dynamics results. The obtained results show that although the actuator surface has a moderate accuracy in calculating parameters such as velocity and pressure, it can predict aerodynamic forces and flow structures with acceptable accuracy. The method presented in this article can be used as an efficient tool in studying more complex cases.

Review History:

Received: Aug. 26, 2021
Revised: Jan. 09, 2021
Accepted: Feb. 21, 2022
Available Online: Mar. 17, 2022

Keywords:

Actuator surface
Airfoil
Steady flow
2D flow
Incompressible flow

1- Introduction

A range of methods for performing airfoil simulations is available which are different in their capability to predict different aspects of aerodynamics behavior. Navier-Stokes simulations with fully-resolved boundaries are now precipitant. However, fully-resolved simulations come at great expense in terms of computational effort, and simulation time especially when multiple airfoils and structures are present in the flow field and complex meshes must be generated. Alternative, less expensive, non-Navier-Stokes models are available, such as momentum theory methods, panel methods, and free wake models. These models suffer from assumptions in their formulation that limit their applicability [1]. It is here that the actuator concept offers the potential for simulating at a less computational cost. In these models, the geometry of blades is represented by volume forces distributed along with disks or lines, or surfaces. In fact, for all actuator disk, surface, or line formulations used in the analysis of airfoil aerodynamics, the surfaces or volumes modeling the airfoil are allowed to be porous to the flow. The purpose of this paper is to investigate the main parameter of the actuator surface technique in airfoil simulation. In this article, the collocated method of Mass Corrected Interpolation Method (MCIM) is utilized for solving two-dimensional unsteady incompressible flow at low Reynolds number [2, 3]. Therefore, the Actuator Surface technique has to be placed in the developed Computational Fluid Dynamics (CFD) solver to be able to predict all parameters quantitatively.

2- Methodology

The two-dimensional Navier-Stokes solver used here is the collocated method of MCIM. The code is based on a control-volume-based finite element method. AS model is included in the CFD solver, as shown in Fig. 1. But before combining this model and CFD solver, it was necessary to develop a CFD solver to add source terms to the momentum equation. The CFD-AS solver described here is applied on unstructured triangular grids. The computational mesh extends 14 chord lengths downstream and 10 chord lengths upstream, above, and below. To determine the accuracy of results, the Actuator Surface method is compared with the Full-CFD simulation method. Besides, a new study is presented to investigate the effect of changing different parameters of the Actuator Surface model on the accuracy of results.

3- Results and Discussion

As mentioned previously, in this study, for the first time, the effects of changing the effective parameters on the Actuator Surface technique and how to select their optimal values are investigated. These parameters include

*Corresponding author’s email: alisadeghi@kntu.ac.ir

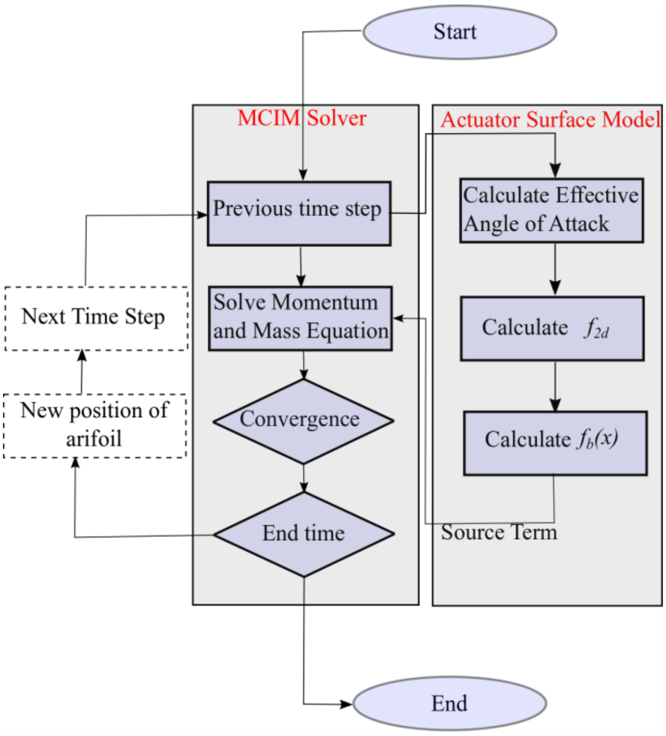

Fig. 1. Process diagram of the Flow solver (combination of MCIM solver and AS).

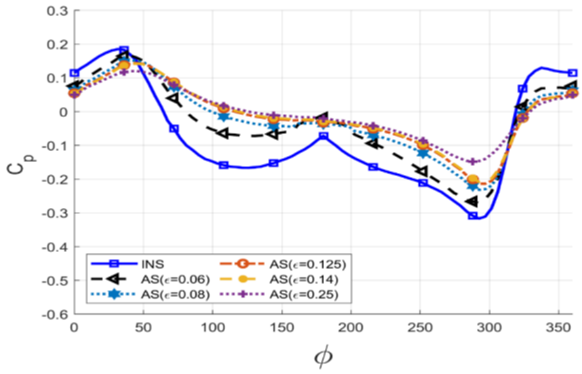

Fig. 2. Comparison of pressure coefficient of CFD-AS for different Gaussian filters for Re=1100 and AOA=8 deg.

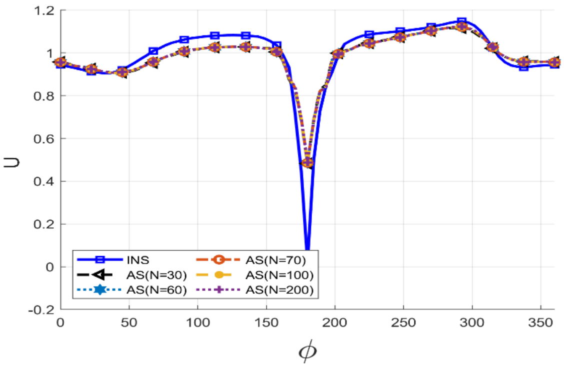

Fig. 3. Comparison of x-velocity of CFD-AS for different density of the sources terms for Re=1100 and AOA=8 deg.

the Gaussian filter, the density of the source terms applied in cells, and the location of the control point. All of these observations are shown in Figs. 2 to 4.

Based on the results, the optimal values for the studied parameters are presented in Table 1.

In Fig. 5, pressure contours and streamlines are shown for both full-CFD and CFD-AS methods. As seen, the streamlines and pressure contours are very similar. It should be noticed that the CFD-AS method does not need a body-fitted mesh. because The geometry of airfoil is represented by volume forces. However, the streamlines do not cross the chord line which the forces are distributed along it.

4- Conclusion

In the present work, an actuator surface model is proposed for the CFD calculation of the flow around an airfoil. This method, called the CFD-AS method requires less computation effort than the full-CFD methods. The obtained results were encouraging. It can be said that the Actuator Surface method, while drastically reducing the computational cost, has

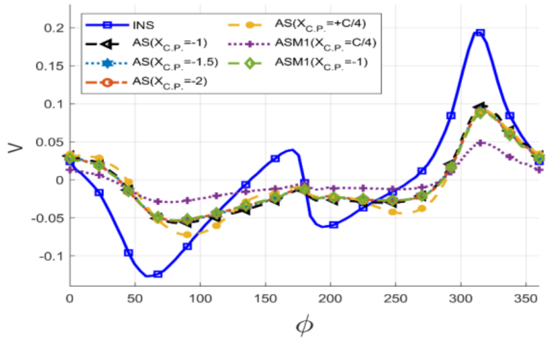

Fig. 4. Comparison of y-velocity of CFD-AS for different locations of control point for Re=1100 and AOA=8 deg.

Table 1. Selected values for examined parameters

Parameter	Value
Gaussian filter (ε)	0.06
The density of the source terms	100
Location of control point	-*C*
Number of cells	13054

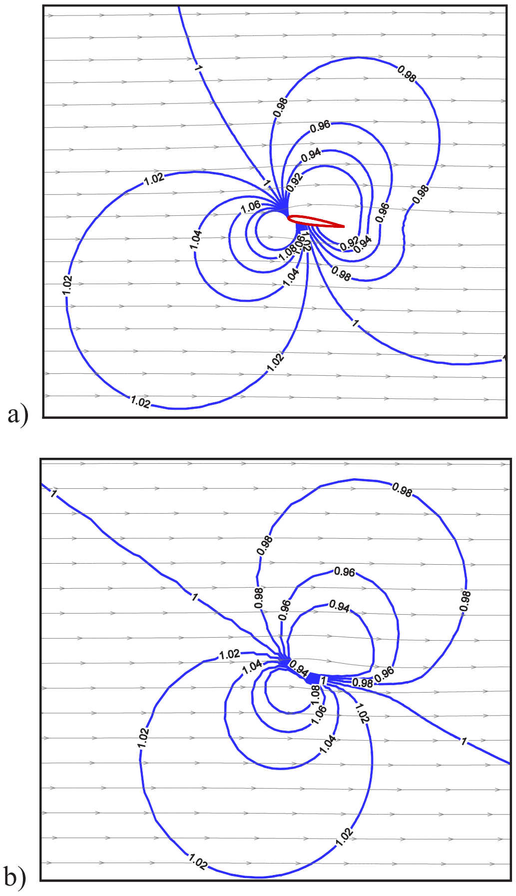

Fig. 5. Streamlines and pressure contours with a)Full-CFD and b)CFD-AS; NACA0015, Re=1100 , Angle of attack of 8 deg.

acceptable accuracy in calculating the flow around the airfoil.

References

[1] D. Linton, G. Barakos, R. Widjaja, B. Thornbern, A new actuator surface model with improved wake model for CFD simulations of rotorcraft, (2017).

[2] H. Alisadeghi, S. Karimian, Comparison of different solution algorithms for collocated method of MCIM to calculate steady and unsteady incompressible flows on unstructured grids, Computers & fluids, 46(1) (2011) 94-100.

[3] H. Alisadeghi, S. Karimian, Different modelings of cell-face velocities and their effects on the pressure–velocity coupling, accuracy and convergence of solution, International Journal for numerical methods in fluids, 65(8) (2011) 969-988.

HOW TO CITE THIS ARTICLE
H. Ettehadi, M. Tebyan, H. Alisadeghi, Simulation of Steady Incompressible Flow around a NACA0015 Airfoil Using Actuator Surface Method and Mass Corrected Interpolation Technique , Amirkabir J. Mech Eng., 54(5) (2022) 217-220.

DOI: 10.22060/mej.2022.20472.7237