

Grid Discretization Study for the Efficient Aerodynamic Analysis of the Very Light Aircraft (VLA) Configuration

Moses Sitio*, Sangho Kim** and Jaewoo Lee***

Department of Aerospace Information Engineering, Konkuk University, 143-701, Seoul, Republic of Korea

Abstract

In this research the development of unstructured grid discretization solution techniques is presented. The purpose is to describe such a conservative discretization scheme applied for experimental validation work. The objective of this paper is to better establish the effects of mesh generation techniques on velocity fields and particle deposition patterns to determine the optimal aerodynamic characteristics.

In order to achieve the objective, the mesh surface discretization approaches used the VLA prototype manufacturing tolerance zone of the outer surface. There were 3 schemes for this discretization study implementation. They are solver validation, grid convergence study and surface tolerance study.

A solver validation work was implemented for the simple 2D and 3D model to get the optimum solver for the VLA model. A grid convergence study was also conducted with a different growth factor and cell spacing, the amount of mesh can be controlled. With several amount of mesh we can get the converged amount of mesh compared to experimental data. The density around surface model can be calculated by controlling the number of element in every important and sensitive surface area of the model.

The solver validation work result provided the optimum solver to employ in the VLA model analysis calculation. The convergence study approach result indicated that the aerodynamic trend characteristic was captured smooth enough compared with the experimental data. During the surface tolerance scheme, it could catch the aerodynamics data of the experiment data. The discretization studies made the validation work more efficient way to achieve the purpose of this paper.

Key words: Grid Discretization, Convergence Study, Surface Discrete, Unstructured Grid

1. Introduction

The Light Aircraft Development Program is an initiative sponsored by the Korean Ministry of Land, Infrastructure and Transport Affairs. The project aims to enhance aircraft design, manufacturing, and certification expertise in the public sector. KLA-100 aircraft development program is being performed by Konkuk University together with 12 other organizations. The program was established in November, 2010 and will conclude in 2014 with the delivery of four prototype aircraft in the Very Light Aircraft (VLA) category [1]. Very Light Aircraft (VLA) category is an aeroplane category with a single engine (spark or compression-ignition) having

not more than two seats, with a Maximum Certificated Take-off Weight of not more than 750 kg and a stalling speed in the landing configuration of not more than 83 km/h (45 knots).

The conceptual and preliminary design phases were allocated to the newly established Korea Aerospace Design, Airworthiness Institute (KADA) at Konkuk University in Seoul [1]. This has offered a unique opportunity to the Aerospace Engineering faculty and students, many of whom are contributing to the development of the aircraft. The detailed design phase as well as the manufacturing will take place at DACC Aerospace, an aerospace composite materials manufacturing company. DACC's business is divided into main areas. They are carbon composites which are suitable for

This is an Open Access article distributed under the terms of the Creative Commons Attribution Non-Commercial License (<http://creativecommons.org/licenses/by-nc/3.0/>) which permits unrestricted non-commercial use, distribution, and reproduction in any medium, provided the original work is properly cited.

© * Master Student
** Professor, Corresponding author: kimsh85@konkuk.ac.kr
*** Professor

use in temperatures up 3,000°C and lightweight structures, which are stronger than steel but lighter than aluminum. The conceptual and preliminary design phases have been completed, resulting in the KLA-100 aircraft concept. The KLA-100 is a two-seat, low wing VLA aircraft with a 620 kg maximum takeoff weight powered by a 100 hp Rotax engine [1]. It is designed for short field performance and a long cruise range.

The conceptual design of light aircraft is typically carried out by using simplified analysis methods and empirical equations in a time consuming, iterative process. Recently, new conceptual design approaches have been proposed that harness computational design optimization methods to reduce time and effort required for the conceptual design while improving the result. Various analysis tools for modeling aerodynamics, weight, stability, and other disciplines are linked in a unified framework. Optimum conceptual designs are obtained by running one of the many widely available optimization algorithms. This approach has been shown to be effective in many case studies in recent literature. However, adoption of these techniques to support real design efforts is virtually non-existent in the light aircraft industry.

The research addresses several issues that arise during the conceptual design phase. Firstly, the reliability of the aircraft weight, drag, and performance analysis methods available early in the design process may be limited. Secondly, aircraft design goals and certification requirements such as stability, stall speed, range, and others need to be considered early in the design process. This paper outlines how these methods can be successfully implemented to support the development of light aircraft. A sophisticated multi-disciplinary light aircraft design and optimization program was developed and used extensively in the conceptual and preliminary design stage of the KLA-100, a new Very Light Aircraft (VLA) currently under development.

The resulting concept continued to satisfy all of the initial requirements when validated using Computational Fluid Dynamics (CFD) and wind tunnel testing. This paper is aimed at investigating the satisfying of the resulting concept scheme by Computational Fluid Dynamics (CFD) works. There will be discretization study to the CFD implementation to prove whether the condition reasonable or not compare to wind tunnel testing. As such it should be able to handle a variety of geometries and to predict the aerodynamic effects of the full body configuration to a reasonable degree of accuracy, while achieving the low computational cost and time needed in design studies. This study considers the manufacturing tolerance effect on the surface area.

In order to conduct the above mentioned research, a clean

model configuration should be taken into consideration. It should be noted that the interaction of flow with the surfaces in computational, as well as the wind tunnel condition. In that respect, the paper will focus only on the aerodynamic effects of the surfaces. This research will serve as a first step towards the understanding and modeling of possible full body Very Light Aircraft computational works.

1.1 Paper objectives

In order to define the scope and setup of the research, a formal objective is formulated. Following the definition of the main objective, a plan can be outlined on how to tackle it and how to structure the study that needs to be performed. The following section explains the objectives of this paper.

The main objective of this paper can be formulated as follows:

Investigate the effect of discretization studies of the full body Very Light Aircraft configuration on the aerodynamics characteristic of a low speed by means of numerical modeling and comparison with wind tunnel test.

The secondary objectives defining the scope of the project are summarized below:

- Studying a tool capable of predicting the aerodynamic effects of full body aircraft configuration
- Establish a range of applicability for the numerical tool
- Employing the solver study and grid convergence study to get the optimum solver and mesh elements for the VLA model analysis
- Focus on surface discrete studies with the VLA manufacturing tolerance baseline in order to get reasonable results compare to wind tunnel data

The numerical tool will consist of a full body aircraft model, combined with an existing design tool, called *FLUENT ANSYS version 14.0*. The incentive for choosing *FLUENT ANSYS version 14.0* will be described in the next chapter.

2. Theoretical

2.1 History and development

Aircraft design begins with the conceptual phase, where possible designs are first imagined and evaluated from initial design requirements. In this phase, the designer has the greatest flexibility in determining the layout and configuration of the aircraft. After the conceptual phase, however, only minor changes to the aircraft configuration may occur. Therefore, it is important to have accurate drag and lift predictions early in the design phase when major

configuration changes can occur. The accuracy of these predictions must be balanced, however, with calculation speed. This is needed so many types of configurations can be compared and so size optimization on a selected configuration may occur.

Aerodynamics for conceptual designs is typically based on linear aerodynamic theory, supplemented with empirical data. These methods work well for subsonic flows, where nonlinearities in the flow are negligible, but break down when the nonlinearities become important. For flows that are entirely supersonic there are nonlinear methods that work well for aerodynamic predictions. However, for transonic flows these methods fail because the flow has both subsonic and supersonic areas. The desire for more accurate lift and drag prediction for transonic flows-along with a more detailed analysis of the flow field for all flows types have resulted in the increased use of Computational Fluid Dynamics (CFD) early in the design stage.

Computational fluid dynamics (CFD) has developed into a valuable design tool. It is now possible to compute the flow around complex aerospace configurations such as complete aircraft, helicopter and spacecraft [2]. This rapid increase in CFD applications over the last few decades has become possible by ever increasing computer power, more efficient numerical algorithms and progress in physical modelling.

2.2 Discretization Study

2.2.1 Grid Convergence Study

The examination of the spatial convergence of a simulation is a straight-forward method for determining the ordered discretization error in a CFD simulation. The method involves performing the simulation on two or more successively finer grids. The term *grid convergence study* is equivalent to the commonly used term *grid refinement study* [3]. As the grid is refined (grid cells become smaller and the number of cells in the flow domain increase) and the time step is refined (reduced) the spatial and temporal discretization errors, respectively, should asymptotically approach zero, excluding computer round-off error [3].

Methods for examining the spatial and temporal convergence of CFD simulations are presented in the book by Roache [4]. They are based on use of Richardson's extrapolation. A general discussion of errors in CFD computations is available for background. We will mostly likely want to determine the error band for the engineering quantities obtained from the finest grid solution. However, if the CFD simulations are part of a design study that may require tens or hundreds of simulations, we may want to use one of the coarser grids. Thus we may also want to be able to

determine the error on the coarser grid.

The easiest approach for generating the series of grids is to generate a grid with what one would consider *fine* grid spacing, perhaps reaching the upper limit of one's tolerance for generating a grid or waiting for the computation on that grid to converge. Then coarser grids can be obtained by removing every other grid line in each coordinate direction. This can be continued to create additional levels of coarser grids. This paper is, therefore, devoted to enhancing the efficiency in computational time and cost, There will be several calculations in several element of mesh to decide the point of mesh to use in next step calculation in discretization works. Levels of coarser grids will be considered in this study. There will be 4 types of elements mesh to consider for convergence study.

2.2.2 Surface Tolerance Study

In Autodesk Simulation CFD, the finite element method is used to reduce the governing partial differential equations (PDEs) to a set of algebraic equations [5]. In this method, the dependent variables are represented by polynomial shape functions over a small area or volume (element) [5]. These representations are substituted into the governing PDEs and then the weighted integral of these equations over the element is taken where the weight function is chosen to be the same as the shape function [5]. The result is a set of algebraic equations for the dependent variable at discrete points or nodes on every element. This paper is, therefore, devoted to enhancing the calculation time; there will be several studies on discrete points or nodes on every element surface as a surface treatment with the automatic surface mesh element control. That scheme will use the VLA tolerance zone as a baseline.

3. Numerical Tools

Design automation with finite element analysis as a simulation and evaluation tool is becoming more and more desired. The ability to do automatic design iteration has constantly been a popular research and engineering topic. Parametric modeling is crucial and necessary for numerical design optimization. However, being able to do parametric modeling does not mean you can use it for optimization. Numerical optimization does have its limitation and assumptions. Our experience has shown that blindly coupling a parametric model together with optimization routine will usually cause serious problems. This is why the above-stated methodology was developed.

ANSYS was one of the solver code that can figure out

above parametric modeling and one of the famous design automation. ANSYS is a finite-element analysis package used widely in industry to simulate the response of a physical system to structural loading, and thermal and electromagnetic effects. ANSYS uses the finite-element method to solve the underlying governing equations and the associated problem-specific boundary conditions.

ANSYS fluid analysis tools include the widely used and well-validated ANSYS FLUENT and ANSYS CFX, available separately or together in the ANSYS CFD bundle [6]. ANSYS FLUENT software contains the broad physical modeling capabilities needed to model flow, turbulence, heat transfer, and reactions for industrial applications ranging from air flow over an aircraft wing to combustion in a furnace, from bubble columns to oil platforms, from blood flow to semiconductor manufacturing, and from clean room design to wastewater treatment plants [6]. Special models that give the software the ability to model in-cylinder combustion, aeroacoustics, turbomachinery, and multiphase systems have served to broaden its reach [6].

This paper is, therefore, devoted to enhancing the calculation and work time efficiency. This paper will implement these ANSYS FLUENT numerical tools. Design automation and friendly using could be the one of the main purpose for choosing these numerical tools. The target for the accuracy was included in this software and it fits with the purpose of this paper.

4. Numerical Analysis

Designs using CFD were based on simplified physical models such as panel methods and linear theory. Where gradient-based optimisation algorithms were employed, aerodynamic sensitivity information was calculated using simple finite-difference techniques. This method of computing design sensitivities requires virtually no modification to the existing analysis code.

In this research, aerodynamic sensitivity is the main feature to be discussed. The importance of this research is with regard to how we manage calculation and sensitivity in a model. The purpose is to obtain accurate data to compare the real condition with the model. The goal is to determine how we attain the condition nearest to the real condition or experimental work. After we have the most accurate data, it can be used as a baseline reference for similar case models, which will be more effective in time and cost compared to experimental work.

In Murayama and Yamamoto computations were performed using two CFD codes based on different mesh

systems (multi-block structured and unstructured mesh) in solving the flow field around a three-element (slat, main, and flap) trapezoidal wing with fuselage [6]. The aerodynamic forces were predicted reasonably, even with the unstructured mesh when moderate settings for the slat and flap were used [7].

Unstructured mesh methods for computational fluid dynamics have been under development for over 25 years [8]. The original attraction of this approach was based on the success achieved in handling complex geometries, as demonstrated through finite-element-based approaches, mainly in the field of structural analysis. In addition to the flexibility in dealing with complex geometries, the ability to easily incorporate adaptive mesh refinement (AMR) strategies also became one of the often quoted advantages of the unstructured mesh approach [8].

For this research, there is also a limitation for acquiring the reasonable data needed compared to experimental data. The limitations in this study are the computer performance, and the unstructured scheme employed. There are two kinds of scheme, which are the main ideas for unstructured mesh generation to approach sensitivity in aerodynamic characteristics, besides other considerations. Every scheme employed contributes differently to reasonable results based on computer performance.

There are three main studies in this discretization scheme. Firstly we conduct the solver study validation. The solver study validation will figure out the effective solver for the KLA 100 model analysis. The second step was the grid element convergence validation. This step performs the optimum element analysis that will use for the next step study validation. The third study was about the surface tolerance study validation. This final study will perform the densities in several important area regarding the manufacturing tolerance zone set.

4.1 Flow Solver

Because of rapid advances in computer speeds and improvements in flow-solvers, a renewed emphasis has been placed on extending computational fluid dynamics (CFD) beyond its traditional role as an analysis tool to design optimisation. The governing equations are *Reynolds-Averaged Navier-Stokes* equations for steady state solutions using unstructured mesh finite volume method. The equations are expressed as a system of conservation laws that relate rate of mass change implies time, momentum, and energy in a control volume of area A to the spatial fluxes of these quantities through the volume. The unstructured grid also employs a surface-based approach.

4.2 Turbulence Model

The fundamental basis of fluid dynamics is the Navier-Stokes equations. In this study, *K-Epsilon Standard Wall Function two-equation model (KES)* will be used to simulate turbulent flow. Meanwhile, in all computation, a full turbulence model is used without transition consideration.

At high Reynolds numbers the rate of dissipation ϵ of kinetic energy is equal to the viscosity multiplied by the fluctuating vortices. An exact transport equation for the fluctuating vortices, and thus the dissipation rate, can be derived from the Navier-Stokes equation. The k-epsilon model consists of the turbulent kinetic energy equation and the dissipation rate equation.

4.3 Discretization Study Validation

4.3.1 Solver study validation

In order to make effectiveness in validation work, it is good to implement simple 2D and 3D model with similar condition and characteristic to the KLA 100 model. These advantages refer to time and cost effectiveness in time calculation. The validation study for several 2D and 3D model are for default solver in computational calculation. It will be more effective if we employ a similar condition to simple model than the complex geometry of this paper model required. The main purpose for this validation study is to minimize the time to find the most effective solver for the KLA-100 model.

4.3.1.1 NACA 64(2)-415 validation

Validation study for this kind of airfoil is quite good. The type of the airfoil used in KLA-100 wing model was similar to this NACA 64(2)-415. It is important to carry out this study, because wing is the most important area for the aerodynamic

phenomenon occurs. For this implementation study, we can hopefully get at least the solver for the phenomena on the wing surface.

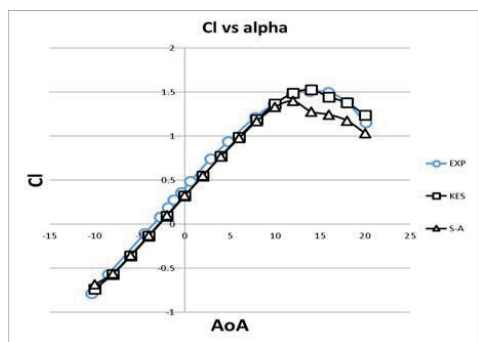
The initial condition for this calculation is according to experiment condition from the book of Theory of Wing Sections [9]. For the subsonic condition and sea-level condition, the 3.1×10^6 Reynolds number is the fit condition for analysis. The results analysis of NACA 64(2)-415 was figured as table 2. K-epsilon and Spalart-Allmaras turbulent model will be employed to these solver validation works.

The K-epsilon and Spalart-Allmaras turbulent model is a low-cost RANS model solving a transport equation for a modified eddy viscosity [10]. It is designed specifically for aerospace applications involving wall-bounded flows. They are the most widely-used engineering turbulence model for industrial applications [10]. These are some considerations for choosing the two kinds of turbulent models as validation work. These are very effective not only in cost and time but also in general user work implementation.

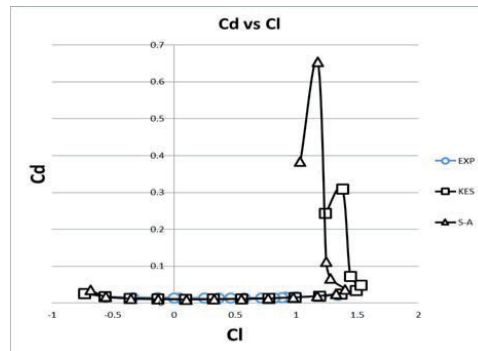
The computational condition was using unstructured

Table 1. Initial condition for the NACA 64(2)-415

Setting case		
Re	3.1×10^6	
ρ (sea-level)	1.225	kg/m ³
μ (sea-level)	1.789×10^{-5}	kg/ms
C	1	m
v	45.25	m/s
Mach	0.133	



(a)



(b)

Fig. 1. (a) Coefficient lift versus angle of attack (b) Coefficient drag versus coefficient lift

mesh with 180,000 mesh elements. The maximum coefficient lift error difference compared to experiment data is about 0.0195. It is quite good to reach the stall condition with the K-Epsilon turbulent model. It can reach the stall area characteristic than Spalart-Allmaras model. For the computational work time, this paper will employ the unstructured scheme for the calculation. It is better for a complex configuration like KLA-100 model. As the comparison, we can see the table result and the graph for the aerodynamic data using K-Epsilon model compare to the experiment data provided in figure 1 and table 2.

For this 2D airfoil validation analysis, we can get the optimum turbulent model for reasonable aerodynamic characteristic approach using K-Epsilon model. The unstructured mesh scheme is quite good also to implement for the model.

4.3.1.2 ONERA M6 wing validation

This case involves the flow over the ONERA M6 wing.

It was tested in a wind tunnel at transonic Mach numbers (0.7, 0.84, 0.88, 0.92) and various angles-of-attack up to 6 degrees. The Reynolds numbers were about 12 million based on the mean aerodynamic chord. The wind tunnel tests are documented by Schmitt and Charpin in the AGARD Report AR-138 published in 1979.

The Onera M6 wing is a classic CFD validation case for external flows because of its simple geometry combined with complexities of transonic flow (i.e. local supersonic flow, shocks, and turbulent boundary layers separation) [11]. It has become a standard for CFD codes because of its inclusion as a validation case in numerous CFD papers over the years. In the proceedings of a single conference, the 14th AIAA CFD Conference (1999), the Onera M6 wing was included in 10 of the approximately 130 papers [11].

Currently, the CFD simulations use the flow field conditions of Test 2308 of Reference 1. Table 3 lists these flow conditions. These correspond to a Reynolds number of 11.72 million based on the mean aerodynamic chord of 0.64607

Table 2. (a) Aerodynamic Computational results (b) Experiment data for NACA 64(2)-415

AoA	Cl	Cd	AoA	Cl	Cl	Cd
-10	-0.735	0.026	20.04	1.153	1.341	0.021
-8	-0.568	0.017	17.93	1.387	1.200	0.018
-6	-0.361	0.013	15.95	1.498	0.997	0.017
-4	-0.135	0.011	13.84	1.509	0.911	0.016
-2	0.092	0.011	11.86	1.477	0.880	0.015
0	0.322	0.011	9.89	1.365	0.819	0.014
2	0.549	0.012	7.78	1.215	0.714	0.014
4	0.772	0.013	4.82	0.941	0.572	0.014
6	0.989	0.015	2.84	0.742	0.461	0.014
8	1.191	0.019	0.62	0.487	0.363	0.013
10	1.365	0.025	-0.36	0.363	0.246	0.013
12	1.488	0.034	-1.22	0.276	0.042	0.014
14	1.529	0.049	-1.84	0.188	-0.00067	0.014
16	1.443	0.073	-2.70	0.083	-0.136	0.015
18	1.378	0.309	-4.43	-0.103	-0.340	0.015
20	1.238	0.243	-8.51	-0.569	-0.562	0.016
			-10.36	-0.787		

(a)

(b)

Table 3. Flow conditions for Test 2308 of Ref. 1. [11]

Mach	Reynolds Number	Angle of Attack(deg)	Angle of sideslip(deg)
0.8395	11.72x10 ⁶	3.06	0.0

meters.

The ONERA M6 wing is a swept, semi-span wing with no twist. It uses a symmetric airfoil using the ONERA D section. The coordinates below indicate that there is a finite thickness to the trailing edge. For CFD simulations, an approximation is usually made of a zero trailing edge thickness.

Comparison data consists of pressure coefficients at sections along the span of the wing obtained in the experiment performed by Schmitt and Charpin as data result figured below. These pressure coefficients are along the lower and upper surfaces of the wing at each of the four sections. The spanwise location of the sections are specified with respect to the wing span. The section location are 20, 44, 65, and 80 percent of wing span measured from the root.

Computational work for this Onera M6 wing was done with the K-epsilon and Spalart-Allmaras turbulent model implementation. With the unstructured scheme, the K-epsilon turbulent model can reach a more reasonable pressure coefficient compare to the experiment data than the Spalart-Allmaras model. The K-epsilon model catches the shock flow better than Spalart-Allmaras as figure 2 below. K-Epsilon turbulent model was a reasonable turbulent

solver also for the low speed condition in order to catch the aerodynamic characteristic. The unstructured scheme was also a reasonable scheme to implement to the KLA-100 model.

4.3.2 Grid Convergence Study validations

The configuration used in this study is a Very Light Aircraft (VLA) 3D full wing-body clean configuration. The fuselage length is 5.8 metres and the wing span is 9.5 metres. The volume is 3.63 m³. It has flap, aileron, horizontal stabiliser, vertical stabiliser, and rudder but only clean configuration was used for this present study. This VLA uses a propeller engine.

Mach number is 0.118 and the Reynolds number, Re, is 0.658x10⁶. In the computations, the flow was assumed to be fully turbulent. The temperature and pressure condition were set at sea level condition for all the cases. The temperature set as 288.16 K and the pressure set as 101,327 Pa. The outer boundary is located 50 chord lengths away from the airfoil. This mesh includes quadrangular elements around all fluid areas and triangular elements to fill all surfaces.

As the theoretical concept approach, these convergence

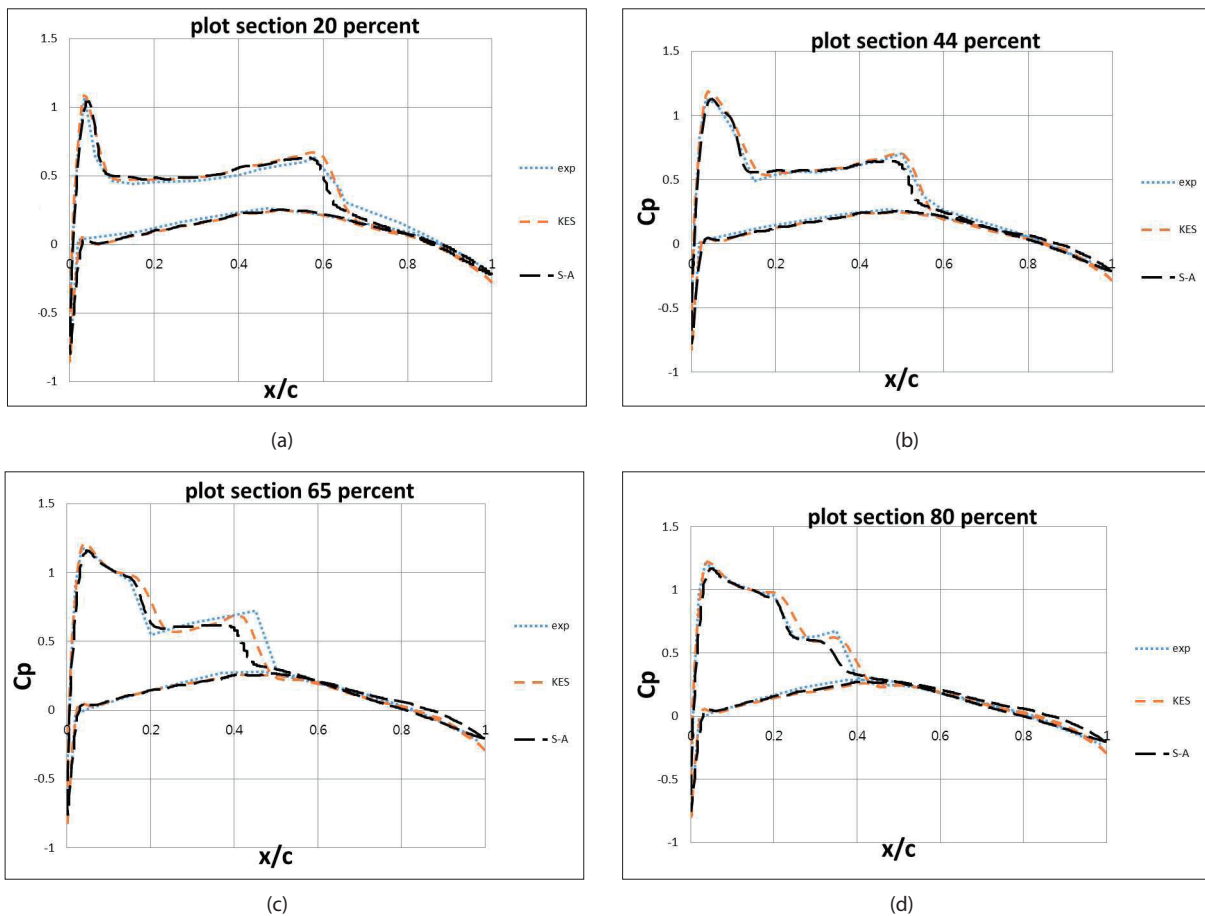


Fig. 2. Plot of the pressure coefficients on the wing surface at section 1(a), section 2(b), section 3(c), and section 4(d) compare to experiment data

studies were done by the validation using several meshes. In Figure 3, it is shown that the changes in the computed lift and drag coefficient decrease monotonically with an increasing grid resolution from 5 million meshes to 8 million meshes. The lift and drag coefficient from 8 million to 12 and 16 million are very similar.

The level of convergence for this large amount of mesh is 10^{-4} with respect to every angle of attack (AoA). The 5, 8, 12 and 16 million mesh elements models need 7.8, 18, 20.52, and 86.4 seconds respectively for one time iteration.

These convergence studies are done to accomplish the optimum mesh element to employ for the discretization study. This is kind of step work to do for discretization scheme. It is difficult to take one element of mesh randomly for discretization work, so that we can approach it with this convergence calculation. After we find the convergence point for this convergence study, it will be used for the baseline element mesh for discretization study.

Table 4. Computed data of CLmax and the Number of Grid Points with Experimental Data

AOA	Element(million)	CL _{max}
10	5	1.29
10	8	1.30
10	12	1.30
10	16	1.30
Experimental data		1.22

The convergence level for the lower angle of attack reaches the 2000 iteration as figure above and the total time was 4.3 hours, 10 hours, 11.25 hours, and 48 hours respectively for 3, 8, 12, and 16 million mesh elements for one angle of attack. The convergence level satisfied in 10-4 for high angle of attack required a 3000 iteration and the total time was 6.5 hours, 15 hours, 17.10 hours, and 72 hours respectively for 3, 8, 12, and 16 million mesh elements for one angle of attack. If we calculate the total time for grid convergence study, for 8 angles of attack, it took 45.4 hours, 105 hours, 119.25 hours, and 504 hours respectively for 5, 8, 12, and 16 million mesh elements. This level of convergence was important for the smooth aerodynamic trend issue. For that issue, it is important to keep the level of satisfaction in every calculation. With that condition, we can get a more reasonable result.

From the Computed data of the maximum coefficient lift and the number of grid points with experimental data above, we can see the converged point of the maximum coefficient lift was around 1.30. Comparing to the CL from experiment data with 1.22, it was still reasonable although our purpose this step study was to find the optimum point of element mesh. The continuous study hopefully gave a more reasonable result for the validation.

4.3.3 Surface Tolerance study validations

The tolerance zone of the outer surface set is considered in this grid discretization as a baseline. Zoning complies with the requirements that apply to the aircraft aerodynamics, depending on the importance, and is divided into the three areas described below:

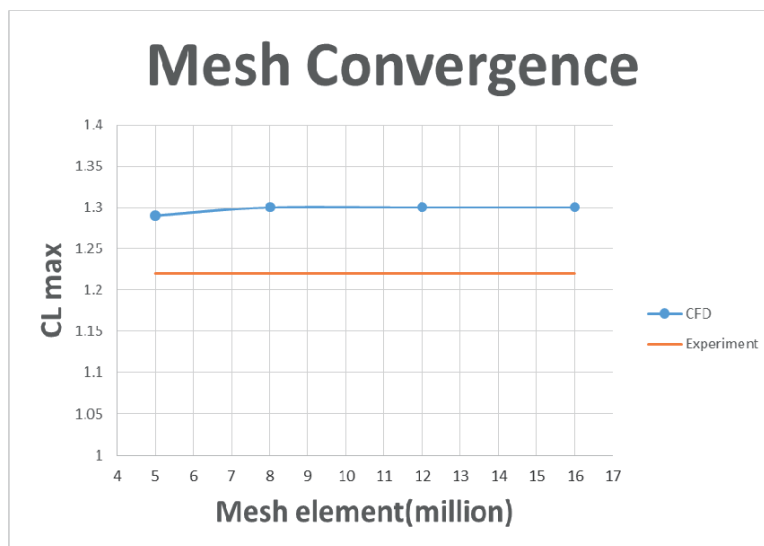





Fig. 3. Comparisons of Computed CLmax and the Number of Grid Points with Experimental Data

-  A type: very important area for consideration of aerodynamic characteristics. Apply the mesh with very high tolerances.
-  B type: important area for consideration of aerodynamic characteristics. Apply the mesh with medium tolerances.
-  C type: average area for consideration of aerodynamic characteristics. Apply the mesh with low tolerances.

Computational works for all the scheme studies use unstructured mesh implementation. As the previous study in convergence study, we already obtained the optimum element mesh to use in discretization study. In this scheme, there will be several surfaces, to show which surface discrete type could be satisfied compare to experiment data.

According to the Prototype Manufacturing Tolerance Zone above as our baseline analysis for the surface discrete, there were 5 main type of area to be considered for surface discrete implementation. Those are wing area discrete, upper wing area discrete, lower wing area discrete, leading & trailing edge area discrete, and fuselage area discrete.

4.4 Results

After the several step and studies were implemented, we obtained the final result for the convergence and discretization study for the determination of the Very Light Aircraft (VLA) manufacturing tolerance as figured below. The results were 8 million mesh elements with the wing, upper wing, lower wing, and TE & LE surface area discrete employed.

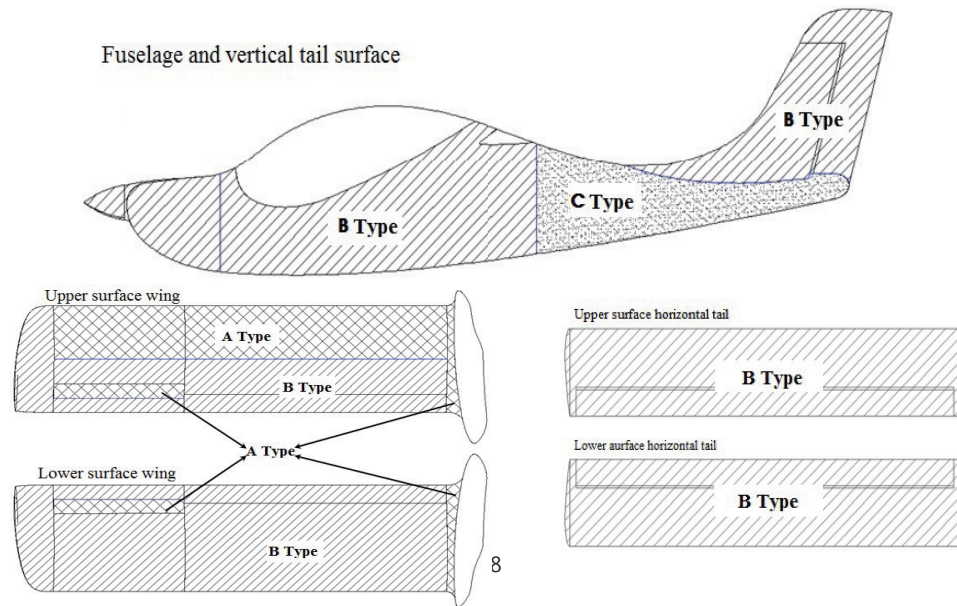


Fig. 4. Prototype Manufacturing Tolerance Zone of the Outer Surface Set [12]

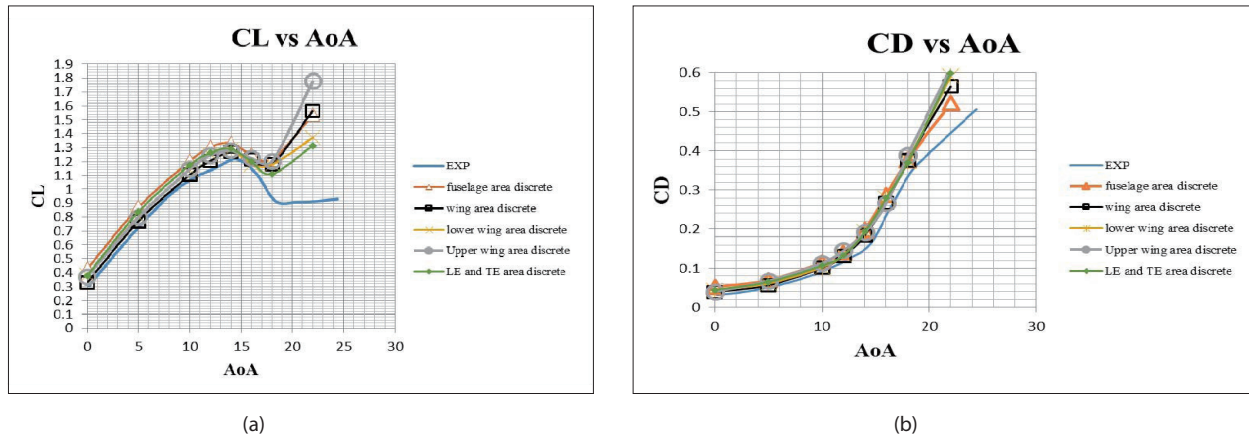


Fig. 5. Comparisons of CL (a) and CD (b) Coefficients versus Angle of Attack for Mesh Tolerance Study

The unstructured mesh with tolerances for several surfaces can indicate the accuracy of the aerodynamic characteristics of the model. As shown in figure 5 and table 5, the indications are that the aerodynamic characteristics approached the required accuracy with a good tolerance on the wing surface. The Calculating time for surface tolerance study validations was 105 hours in every discrete scheme. It was similar to the 8 million mesh elements time calculation from grid convergence study. This was similar because we use the same mesh element for this discrete scheme. The total time for all the discrete scheme was 525 hours.

5. Conclusions and Recommendations

5.1 Conclusions

In grid convergence study using unstructured meshes, the 8 million element mesh was found to be the optimum mesh for the aerodynamic sensitivities of the KLA 100 configuration. The grid discretization study has been performed to investigate the time and cost effectiveness of the same size mesh with various local surface mesh densities.

The feasibility of the proposed process and the numerical results for this type of aircraft have been presented. The results of the grid discretization study confirmed that the current prototype manufacturing tolerance as our baseline was, in general, properly set. The lower wing surface tolerance should be equivalent to the upper wing surface tolerance. The wing surface is the main important area for consideration in order to achieve aerodynamic characteristic validation. The most feasible study for this KLA-100 model was employing the K-epsilon standard turbulent model with 8 million mesh elements and with wing area discrete implementation.

5.2 Recommendations

There are several recommendations for future works regarding this research. These are kinds of advanced work to

do for better achievement in this research field. The future works can be explained as below.

- 1) Improved ways of grid density localization for grid discretization study.
- 2) More refined grid discretization study for aerodynamic sensitivities of the control surface.
- 3) Corresponding intensive aerodynamic analysis for KLA 100 configurations with the various deflection angles of the control surfaces such as flap, aileron, elevator, and rudder.
- 4) With this reasonable result, we can use these results for the Fluid Structure Interaction works for the aerodynamic data needed. Even in this future work there was a kind of issue to be considered regarding both different target and consideration in mesh discretization from aerodynamic and structure works, this will be an interesting topic research to continue in future research.

Acknowledgment

This research was supported by a grant (1615001723) from the Light Aircraft Development Program funded the Ministry of Land, Infrastructure and Transport affairs of the Korean government. The authors appreciate the financial support.

References

- [1] URL: http://www.casi.ca/assets/development_and_application_of_mdo_tool_for_light_sport_aircraft_design_lee_nefeld.pdf
- [2] URL: <http://www.transhumanist.com/volume1/moravec.htm>
- [3] URL: <http://www.grc.nasa.gov/WWW/wind/valid/tutorial/spatconv.html>
- [4] Roache, P. J., *Verification and Validation in Computational Science and Engineering*, Hermosa Publishers, Albuquerque, New Mexico, 1998.

Table 5. (a) Maximum CL in every discrete scheme (b) CD data from the maximum CL

Discrete	AoA	CLmax
Wing	14	1.266
Fuselage	14	1.333
Lower wing	14	1.289
Upper wing	14	1.279
LE and TE	14	1.294
Experiment	14	1.217

(a)

Discrete	AoA	CD _{CLmax}
Wing	14	0.190
Fuselage	14	0.197
Lower wing	14	0.193
Upper wing	14	0.191
LE and TE	14	0.192
Experiment	14	0.162

(b)

[5] URL: http://wikihelp.autodesk.com/Simulation_CFD/enu/2014/Help/0360Learning360/0481-Referenc481/0503-Theoreti503/0526-Discreti526

[6] URL: <http://www.ansys.com/Products/Simulation+Technology/Fluid+Dynamics/Fluid+Dynamics+Products/ANSYS+Fluent>

[7] Lei, Z., "Grid Convergence Study for High-Lift Devices: Three-Element Airfoil", *West-East High Speed Flow Field Conference*, Moscow, Russia, 2007.

[8] Mavriplis, J.D., "Unstructured Mesh Discretizations and Solvers for Computational Aerodynamics", *18th AIAA Computational Fluid Dynamics Conference*, Miami, Florida, 2007.

[9] Abbott, I.H., and Doenhoff A.E.V, *Theory of Wing Sections: Including a Summary of Airfoil Data*, 1959.

[10] ANSYS Training slide presentation, Turbulent Model, 2009.

[11] URL: <http://www.grc.nasa.gov/WWW/wind/valid/m6wing/m6wing.html>

[12] Deck Aviation Institute consultation meeting for KLA-100 tolerance (Structural Design), 2012.

[13] Dimchev, M., *Experimental and Numerical Study on Wingtip Mounted Propellers for Low Aspect Ratio UAV Design*, Master of Science Thesis, Delft University of Technology, Delft, 2012.

[14] Lucas, P., *Efficient Numerical Methods to Compute Unsteady Subsonic Flows on Unstructured Grids*, Philosophy Doctor Dissertation, Delft University of Technology, Delft, 2010.

[15] Jansen, D.P., *Passive Flow Separation Control on an Airfoil-Flap Model-The Effect of Cylinders and Vortex Generators*, Master of Science Thesis, Delft University of Technology, Delft, 2012.

[16] ANSYS FLUENT brochure version 14.0, The ANSYS Fluent package combines deep physics and years of simulation development expertise to solve CFD challenges-right out of the box, 2011.

[17] URL: <http://ansys.net/ansys/papers/Ans-Mag1b.pdf>

[18] Sports-class light aircraft development - wind tunnel tests (Contract Research) Report, Air Force Academy, Aerospace Engineering and medium-sized wind tunnel laboratory. 2011.