Utilization of CFD for the aerodynamic analysis of a subsonic rocket

Hesaplamalı Akışkanlar Dinamiği ile ses altı bir roketin aerodinamik analizi

Yazarlar (Authors): Zeynep AYTAÇ¹, Fatih AKTAŞ²

ORCID¹: 0000-0003-0717-5287
ORCID²: 0000-0002-1594-5002

Bu makaleye şu şekilde atıfta bulunabilirsiniz (To cite to this article): Aytaç Z. ve Aktaş F., “Utilization of CFD for the aerodynamic analysis of a subsonic rocket”, Politeknik Dergisi, 23(3): 879-887, (2020).

Erişim linki (To link to this article): http://dergipark.org.tr/politeknik/archive

DOI: 10.2339/politeknik.711003
Utilization of CFD for the Aerodynamic Analysis of a Subsonic Rocket

Highlights
- The design and analyses of a subsonic rocket was carried out with the utilization of CFD.
- The effects of several critical parameters on the rocket performance were investigated.
- An increment in Mach number at approximately 30% results in an increment of drag coefficient nearly 68%.
- Changing the turbulence intensity does not make any significant difference on drag coefficient.
- The drag coefficient obtained from k-ω is higher than that of obtained from k-ω SST.

Graphical Abstract
The design process of a rocket with experimental processes and measuring all the necessary variables in wind tunnels can be exhausting, time and money consuming for most researchers. A reasonable prediction of these parameters with the utilization of appropriate approaches is offered by CFD simulations. In the present study, traditional CFD methodology was followed in order to simplify the design process.

Figure A. Dimensions of the designed rocket
Figure B. The design methodology

Aim
Nowadays, every single country aims to have a domestic and national defense industry. In accordance with this purpose, the design of missile structures has become more important than ever. In this study, the design and analyses of a subsonic rocket was carried out with the utilization of Computational Fluid Dynamics (CFD) tools. Also, the effects of several critical parameters; i.e. Mach number, turbulence intensity, turbulence model, on the rocket performance were investigated.

Design & Methodology
Initially, the 3D model of the missile was created using CATIA in the light of dimensional specifications. After the geometry generation, the 3D model of the rocket was meshed using ANSYS Meshing. A high-quality mesh is critical to provide the accuracy and stability of a numerical solution. So, the grid structure should be constructed neatly in order to successfully represent the physical phenomena in the flow domain. In order to ensure that the results are independent of the mesh structure, a mesh independency study was carried out. Afterwards, ANSYS CFX-Pre is used to set up the cases and ANSYS CFX Solver is used to simulate the problem.

Originality
This study represents the design process of a subsonic rocket and investigates the effect of the parameters used in CFD analyses. There is not a similar conducted study in literature representing both.

Findings
An increment in Mach number at approximately 30% results in an increment of drag coefficient nearly 68% and although the appropriate turbulence intensity should be used for every unique problem, in this case, this parameter is not a critical variable to ponder upon. Moreover, the turbulence model has a substantial effect on the obtained results; so, the utilization of the appropriate model is crucial.

Conclusion
CFD tools are sufficient for the prediction of the flow around a subsonic rocket. The key point in the design process is to set up the case appropriately.

Declaration of Ethical Standards
The author(s) of this article declare that the materials and methods used in this study do not require ethical committee permission and/or legal-special permission.
Hesaplamalı Akışkanlar Dinamigi ile Ses Altı Bir Roketin Aerodinamik Analizi

 Araştırma Makalesi / Research Article

Zeynep AYTAÇ*, Fatih AKTAŞ

Mühendislik Fakültesi, Makine Müh. Bölümü, Gazi Üniversitesi, Ankara, Türkiye

(Geliş/Received : 29.03.2020 ; Kabul/Accepted : 07.04.2020)

ÖZ

 Günümüzde her ülkenin, kendi yerli ve milli savunma sanayisini geliştirmeye hedeflediği durumda, bu doğrultuda, füze ve roket gibi iki önemli proje olarak gün yüzüne çıkmaktadır. Bu doğrultuda, Hesaplamalı Akışkanlar Dinamiği (HAD) yardımıyla, ses altı huzurarophy, roket tasarım analizleri ve aerodinamik analizleri gerçekleştirilmektedir. Ayrıca, Mach sayısı, türbülans yoğunluğu ve türbülans modeli gibi kritik parametrelere göre roket performansına etkileri incelenmektedir. Çalışma sonucunda, Mach sayısındaki ciddi bir etkiye sahip olduğu görülmüştür. Mach sayısındaki %30'lu bir artış, sürüklenme katsayısının yaklaşık %68 artmasına sebep olmuştur. Bu durum, türbülans yoğunluğunun değiştiğinin ise sürüklenme katsayısında belirgin bir farka sebep olmadığını göstermiştir. Her ne kadar, her problem için uygun türbülans yoğunluğunun seçilmesinin önemli olduğu bilinse de, mevcut problem için türbülans yoğunluğu seçimini zaman harcanacak bir kriter olmadığı sonucuna varılmıştır. Son olarak, türbülans modeli seçiminde, beklendiği gibi, tasarım açısından oldukça önemlidir. Bu nedenle, çözümler için literatürde yaygın olarak kullanılan k-ω SST ve diğer bir model olan k-ω modeli ve diğer bir model olan k-ω SST modeli ile gerçekleştirdikleri çalışmadan, sürüklenme katsayısı açısından yaklaşık %12 fark olduğu görülmüştür. Beklendiği gibi, k-ω modelinden elde edilen sonuç, k-ω SST modelinden elde edilen sonuçta daha yüksek alır.

Anahtar Kelimeler: Roket, Hesaplamalı Akışkanlar Dinamiği, tasarım metodolojisi, ses altı akış, dış akış.

Utilization of CFD for the Aerodynamic Analysis of a Subsonic Rocket

ABSTRACT

Nowadays, every single country aims to have a domestic and national defense industry. In accordance with this purpose, the design of missile structures has become more important than ever. In this study, the design and analyses of a subsonic rocket was carried out with the utilization of Computational Fluid Dynamics (CFD) tools. Also, the effects of several critical parameters; i.e. Mach number, turbulence intensity, turbulence model, on the rocket performance were investigated. It was found out that a variation in Mach number has a substantial effect on the drag coefficient; i.e. an increment in Mach number at approximately 30% results in an increment of drag coefficient nearly 68%. Contrarily, changing the turbulence intensity does not make any significant difference on drag coefficient. Although the appropriate turbulence intensity should be used for every unique problem, in this case, this parameter is not a critical variable to ponder upon. Finally, the implementation of the appropriate turbulence model is critical in the design process as expected. Utilization of k-ω and k-ω SST models differs approximately 12% in terms of drag coefficient; the drag coefficient obtained from k-ω is higher than that of obtained from k-ω SST.

Keywords: Rocket, Computational Fluid Dynamics, design methodology, subsonic flow, external flow.

1. INTRODUCTION

Today, with the developing political strategies and relationships, each country attaches particular importance to their defense industry. Similarly, Turkey aims to design and manufacture its own missiles. In accordance with this purpose, the knowledge of the design process of these structures started to develop and became widespread than ever.

Rockets are used for various purposes in defense and research industry. They carry payloads into the orbit or space, or they can be used for weapon applications. The first rocket in history is designed and manufactured in China in 1200 and used as fireworks during the New Year celebrations [1]. A body immersed in a fluid medium

*Sorumlu Yazar (Corresponding Author)
E-posta : fsaytac@gmail.com.tr

879
the internals are parachutes and shock cord, electronic accessories and motor. Figure 1 represents the structure of a rocket.

![Figure 1. The rocket structure [4]](image_url)

The nosecone splits the airflow around the vehicle which maintains the speed and each nosecone has a unique structure designed for that specific airflow and vehicle. The amount of air resistance that the vehicle experiences depends mainly on the shape of the nose cone, the body diameter and the speed. For subsonic applications, it is known that a rounded curved nosecone shape is more beneficial. The body cylinder keeps the pressure distribution even throughout the vehicle and it constitutes the main structure of the rocket. The larger the diameter of the body gets, the more drag force that the vehicle is exposed to. In addition, it provides a safe housing for the internal components. The fins are required for the stability of the vehicle, even if they cause the drag force to increase. These components should be designed optimally to fulfil the mission successfully [5].

The airflow characteristics such as airflow velocity, flow rate, pressure, drag force, etc. affect substantially the exterior ballistics of the rocket [8]. The aerodynamic coefficients, which are drag and lift coefficients, are dimensionless quantities which are used to determine the aerodynamic characteristics of a structure. They are determined by the ratio of several forces, rather than just the forces themselves. The aerodynamic forces result primarily from the differences in pressure and viscous shearing stresses [2].

The drag coefficient of a structure is used to model the drag of a body immersed in a fluid medium. The drag coefficient is the most critical parameter for the investigation of exterior ballistics. Consequently, the rocket engine thrust characteristics are directly influenced by this specific parameter. It depends on the shape of the structure, inclination and the flow condition and it is expressed with Equation 1.

$$C_D = \frac{2F_D}{\rho AV^2}$$  \hspace{1cm} (1)

Here, $C_D$ presents the drag coefficient, $F_D$ represents the drag force, $\rho$ the density, $A$ the cross-sectional area of the body and $V$ the speed. As the drag coefficient gets smaller, one can understand that the structure experiences a less aerodynamic drag.

Similarly, the lift coefficient expresses the ratio of the lift force to the force resulting from the multiplication of dynamic pressure to the area. Lift force is the force which is perpendicular to the oncoming flow direction. For a lift force to be generated, a pressure difference between the upper and lower sides of the structure is required.

1.1. Utilization of CFD for Aerodynamic Design of a Rocket

The design process of a rocket with experimental processes and measuring all the necessary variables in wind tunnels can be exhausting, time and money consuming for most researchers [9]. A reasonable prediction of these parameters with the utilization of appropriate approaches is offered by CFD simulations. Today, with the advances and conveniences in computer technology and computational tools led CFD to become an essential design tool, reducing the costs of experimental studies [10]. CFD tools enable accurate solutions to complex, three-dimensional problems for missile aerodynamics [11]. When the problem is formed using the right numerical models and approaches, CFD offers qualified information that can be derived routinely for a wide range of applications [12]. It is widely used in aeronautical applications during the conceptual and preliminary design stages, as it reduces the design cycle time and minimizes the expenses related with the experimental procedures [13,14,15].

1.2. Specifications of the Rocket

The dimensions of the designed rocket are given in Figure 2. The specified dimensions are given in millimeters.
In addition, the rocket is required to operate at atmospheric conditions (1 atm, 25°C) with a maximum velocity of 170 m/s.

2. METHODOLOGY

Each missile structure has a unique design depending on its requirements. As mentioned above, as the experimental process is infeasible in many ways, Computational Fluid Dynamics becomes a prominent tool at this point. In the present study, traditional CFD methodology was followed in order to simplify the design process. The outline is given in Figure 3.

2.1. 3D Modeling

Initially, the 3D model of the missile was created using CATIA in the light of dimensional specifications. The generated model is given in Figure 4.

2.2. Grid Generation

After the geometry generation, the 3D model of the rocket was meshed using ANSYS Meshing. A high-quality mesh is critical to provide the accuracy and stability of a numerical solution [2]. So, the grid structure should be constructed neatly in order to successfully represent the physical phenomena in the flow domain. Here, it is known that the boundary layer resolution at the top of the body is of substantial importance. In addition, inlet and outlet regions constitute the other locations to pay attention on. Also, as this problem requires the determination of drag force, the boundary layer is required to have a fine mesh structure.
The generated grid structure is given in Figure 5.

As it can be seen from Figure 5, the region surrounding the rocket profile has a denser mesh structure, whereas the density of the mesh is decreasing as one progresses through the boundary of the domain. This structure is constructed by using inflation layers. The first layer thickness of the inflation layer is 0.0018 mm. This parameter is kept as small as possible so that the first layer is located close to the rocket body [8]. It is mentioned above that this structure is necessary in order to successfully resolving the boundary layer.

The detail of mesh statistics is given in Figure 6.

The skewness value represents the difference between the shape of a cell and the shape of an equilateral cell of equivalent volume. So, this value is needed to be minimized in fine mesh structures. A general rule for most flows that the skewness is below 0.95, with an average value of much lower [2]. The generated structure in this study has a maximum skewness of 0.9 and an average skewness of 0.17344. In other words, the grid structure stays on the safe side.

Figure 7 represents the mesh metrics in terms of skewness.

It can be seen from Figure 5 that most of the elements have a skewness of 0.5 or lower. Only a minority of them have a skewness value larger than 0.5, which are the ones located near the fins. This is a result of the sharp corners and edges of the fin profile, obstructing the smooth transition of the mesh cells.

Finally, in order to ensure that the results are independent of the mesh structure, a mesh independency study was carried out. The results are compared with respect to the drag force calculated. The results are given in Table 1.

| Element Number [$10^3$] | Drag Force [N] |
|-------------------------|----------------|
| 13500                   | 77.2           |
| 10100                   | 77.05          |
| 9200                    | 77.06          |
| 5800                    | 78.48          |
| 5600                    | 78.65          |
| 1600                    | 78.9           |
| 635                     | 84.02          |

Even decreasing the mesh number by half; from 13.5M to 5.8M, does not create a difference more than 2%. So, rather than using 13.5M elements, it is more feasible to
use 5.8M elements to save from computational time and resources.

2.3. Solver Settings and CFD Analyses
To model the 3D motion of a fluid particle, Navies-Stokes equations are used. The equations are given in the following subsections.

2.3.1. Conservation of Mass
The mass conservation for a particle having dimensions of dx, dy and dz is expressed with Equation 2.1.

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0 \tag{2.1}
\]

x - component:

\[
\rho \frac{D\bar{u}}{Dt} = \rho \left[ \frac{\partial}{\partial y} (\bar{u}^2) + \frac{\partial}{\partial z} (\bar{w} \bar{u}) \right] = \rho g_x - \frac{\partial \bar{p}}{\partial x} + \frac{\partial}{\partial x} \left[ \mu \frac{\partial \bar{u}}{\partial x} - \rho \bar{u}^2 \right] + \frac{\partial}{\partial y} \left[ \mu \frac{\partial \bar{u}}{\partial y} - \rho \bar{u} \bar{v} \right] + \frac{\partial}{\partial z} \left[ \mu \frac{\partial \bar{u}}{\partial z} - \rho \bar{u} \bar{w} \right]
\]

y - component:

\[
\rho \frac{D\bar{v}}{Dt} = \rho \left[ \frac{\partial}{\partial x} (\bar{u} \bar{v}) + \frac{\partial}{\partial z} (\bar{w} \bar{v}) \right] = \rho g_y - \frac{\partial \bar{p}}{\partial y} + \frac{\partial}{\partial x} \left[ \mu \frac{\partial \bar{v}}{\partial x} - \rho \bar{u} \bar{v} \right] + \frac{\partial}{\partial y} \left[ \mu \frac{\partial \bar{v}}{\partial y} - \rho \bar{v}^2 \right] + \frac{\partial}{\partial z} \left[ \mu \frac{\partial \bar{v}}{\partial z} - \rho \bar{v} \bar{w} \right]
\]

z – component:

\[
\rho \frac{D\bar{w}}{Dt} = \rho \left[ \frac{\partial}{\partial x} (\bar{u} \bar{w}) + \frac{\partial}{\partial y} (\bar{v} \bar{w}) \right] = \rho g_z - \frac{\partial \bar{p}}{\partial z} + \frac{\partial}{\partial x} \left[ \mu \frac{\partial \bar{w}}{\partial x} - \rho \bar{u} \bar{w} \right] + \frac{\partial}{\partial y} \left[ \mu \frac{\partial \bar{w}}{\partial y} - \rho \bar{v} \bar{w} \right] + \frac{\partial}{\partial z} \left[ \mu \frac{\partial \bar{w}}{\partial z} - \rho \bar{w}^2 \right]
\]

2.3.2. Conservation of Momentum
Law of conservation of momentum is simply the Newton’s second law of motion. It states that the time rate of change of momentum of a system is equal to the sum of external forces acting on that body and is expressed with Equation 2.2.

\[
\frac{d(\rho u_i)}{dt} + \frac{\partial \rho u_i u_j}{\partial x_j} = -\frac{\partial p}{\rho \partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_i^2} + F_i \tag{2.2}
\]

The external flow domain of a rocket is simulated using Reynolds Averaged Navier – Stokes (RANS) equations. RANS methods are widely used in industrial applications [16]. The equations are given in Equation 2.3, separately for x, y and z axes.

2.3.3. Turbulence Model
Although the Navier-Stokes equations are simplified with the conservation equations and the averaging procedure, it is still not possible to solve them analytically. So, the two-equations coming from the turbulence model are required in order to solve the flow accurately. The present study uses k-\omega SST model in addition to RANS equations. This model is the most suitable model for aeronautics applications where strong adverse pressure gradients and separation are observed. Although standard k – \omega model over predicts separation, k – \omega SST comes through this problem. The utilization of k-\omega SST makes the model directly usable from the boundary layer region all the way down through the viscous sublayer. This formulation switches to k-\epsilon behavior within the free stream; overcoming the over predicting model.

2.4. Simulation
ANSYS CFX-Pre is used to set up the cases and ANSYS CFX Solver is used to simulate the problem. The regions used to define the boundary conditions are given in Figure 8.

In Figure 8, the pink region represents the inlet, the blue represents the outlet, the green parts represent the walls. The yellow part is the rocket. The inlet and outlet locations are selected as “Inlet” and “Outlet” boundary types, respectively. The inlet boundary is defined with the normal speed, 170 m/s and the outlet boundary is defined with the static pressure, 0 Pa. The reference pressure is selected as 1 atm. The rocket is defined as a...
non-slip wall and the remaining regions are selected as symmetry. The fluid is air – ideal gas and as mentioned, k-ω SST turbulence model is utilized. The residual target is specified as 10^{-6}. Furthermore, a monitor point was used to monitor the velocity value in the middle section to check the convergence. The problem was solved using steady-state conditions.

After setting up the cases, analyses were conducted using CFX Solver Manager. The obtained results are given in Section 3.

3. RESULTS AND EVALUATION
The analyses were conducted for various Mach numbers of 0.35, 0.5 and 0.65. Also, results from several turbulence models and turbulence intensities were compared with each other. The results are given separately for each parametric analysis. The actually designed case is the one with the low intensity turbulence, 0.5 Ma and k-ω SST turbulence model.

3.1. The Design Case
The contours of pressure, total pressure, velocity, y+ and the velocity vectors are given in the proceeding figures.

As 170 m/s corresponds to 0.5 Ma, the enclosure region contour is red in color. According to the flow separation at the nose cone, a region of low velocity and a stagnation point is observed, and a thin boundary layer is developed at the top of the body region. At the outlet region, a region of low velocities and even zero velocity is observed. This is due to the vortex formations at the outlet, which results in flow circulations locally.
From Figure 11, it can be seen that the velocity contours exhibit a homogenous direction, through the inlet to the outlet. The developed boundary layer can be seen more clearly, and the recirculation region at the back region of the rocket is obvious. In this location, because of the recirculation and formed vortexes, the velocity decreases substantially.

The y+ value is critical in terms of accuracy of the solution. Each turbulence model requires a different range y+ values to attain a reliable solution. y+ simply defines the dimensionless distance from the wall which is used to check the location of the first node away from the wall [2]. As it depends on the mesh structure, it has a significant effect on the model’s ability to solve the boundary layer. For the current turbulence model, this value needs to be 1 or smaller. From Figure 12, it can be seen that the maximum value of y+ is 1.0, which are located on the fins as they have sharp edges and corners. Throughout the rest of the body, it is 0.7 or smaller; which expresses that the obtained results are accurate enough to resolve the boundary layer.

As the Mach number is increased, it can be seen that the drag force increases relatively. This is an expected result, since the increment in Mach number is equal to the increment in velocity. An increased velocity corresponds to an increased drag force, due to the increased frictional forces. An increment in Mach number at approximately 30% results in an increment of drag coefficient nearly 68%.

3.2.2. Turbulence Intensity

Turbulence intensity represents the turbulence level of the flow. It is determined depending on the previous experience on the designer and the state-of-art. Generally high turbulence level is used in high speed flows in complex geometries; such as turbomachines. The turbulence intensity is between 5% and 20% for high intensity. Medium intensity is the most common used level, as it is used for flows in not-so-complex devices or low speed flows. Its intensity varies between 1% and 5%. Low intensity is used for flows originating from a fluid which is not moving, e.g. external flow across cars, submarines and aircrafts. Low intensity has a turbulence
level lower than 1%. As this problem is typically an external flow around an air vehicle, low intensity level is used.

The results between the turbulence levels are given in Figure 15.

![Figure 15. Turbulence intensity vs. drag coefficient](image)

As expected, drag force increases with the increased turbulence level. However, the variation is between 77 and 77.38; which will not make quite a difference in the design process. Therefore, although the appropriate turbulence intensity should be used for every unique problem, in this case, this parameter is not a critical variable to ponder upon.

3.2.3. Turbulence Model

The specified turbulence model determines the two equations which will be solved with the RANS equations. As mentioned before, k-ω SST model was used for the design process of this case. However, the effects of several turbulence models were investigated, and the results are given in Table 2.

| Turbulence Model | Drag Force [N] | Drag Coefficient |
|------------------|----------------|------------------|
| BSL Reynolds     | 81.7362        | 0.452837         |
| k-ε             | 79.6909        | 0.441067         |
| k-ω             | 86.7081        | 0.479905         |
| RNG Epsilon     | 78.9961        | 0.43722          |
| k-ω SST         | 77.0578        | 0.42704          |

Accepting the result obtained from k-ω SST model as a reference, it can be seen that RNG Epsilon model is the nearest, followed by k-ε, BSL Reynolds and finally k-ω. One can expect k-ω model to give the nearest result to the k-ω SST model, however, k-ω model overpredicts the separation in the boundary layer, which in turn affects the drag force in a substantial manner.

4. CONCLUSION

The present study involves the design of a rocket in a subsonic speed with the utilization of CFD tools and investigates the effect of several critical parameters on the drag force; which is a reference result in rocket design processes.

It is obtained that the drag force is extremely sensitive to the variations in Mach number. A 30% increment in Mach number resulted in an increment of drag coefficient by nearly 70%.

The three turbulence intensity options existing in ANSYS CFX was used to obtain the differences in drag forces. As mentioned before, although every designer should use the correct turbulence intensity for each design problem, this case does not reveal a distinct difference between each intensity.

Finally, the available turbulence models were used to analyze the resulting drag forces. As the k-ω SST model is taken as a reference, RNG-Epsilon model gives the closest result to that of k-ω SST, then comes the k-ε, BSL Reynolds and k-ω models. Again, each problem has its own unique specifications and flow characteristics and turbulence model should be decided in the light of these requirements. Taking into consideration that the k-ε, k-ω and k-ω SST models are the most common ones used in commercial CFD applications, the designer should keep in sight that the turbulence model affects the obtained results substantially.

REFERENCES

[1] Howell, E., “Rockets: A History”, space.com contributor, (2015).
[2] Hammargren, K., “Aerodynamics Modeling of Sounding Rockets”, Ms. Thesis, Lulea University of Technology, (2018).
[3] Guzelbey, I.H., Sumnu, A. and Dogru, M.H., “A Review of Aerodynamic Shape Optimization for a Missile”, The Eurasia Proceedings of Science, Technology, Engineering & Mathematics (EPSTEM), 4: 94-102, (2018).
[4] https://cpb-us-w2.wpmucdn.com/u.osu.edu/dist/h/38251/files/2018/01/Workshop-1-Aero-and-Propulsion-qix91h.pdf, Aerodynamics and Propulsion, Buckeye Space Launch Initiative.
[5] Cronvich, L.L., “Missile Aerodynamics”, John Hopkins APL Technical Digest, 175-186, (1983).
[6] Gönç, L.O., “Computation of External Flow Around Rotating Bodies”, PhD Thesis, Middle East Technical University, (2005).
[7] Başoğlu, O., “Three Dimensional Aerodynamic Analysis of Missiles by a Panel Method”, MS Thesis, Middle East Technical University, (2002).
[8] Fedaravičius, A., Kilkhevičius, S., Survila, A. and Patašienė, L., “Analysis of Aerodynamic Characteristics of the Rocket-Target for the “Stinger” System”, Problems of Mechatronics Armament, Aviation, Safety Engineering, 7, 1(23): 7-16, (2016).
[9] Lopez, D., Dominguez, D. and Gonzalo, J., “Impact of Turbulence Modeling on External Supersonic Flow Field Simulations in Rocket Aerodynamics”, *International Journal of Computational Fluid Dynamics*, 27(8-10): 332-341.

[10] Elliot, J. and Peraire, J., “Practical 3-D Aerodynamic Design and Optimization Using Unstructured Meshes”, *AIAA Journal*, 35(9): 1479-1485, (1997).

[11] Sahu, J. and Heavey, K.R., “Parallel CFD Computations of Projectile Aerodynamics with a Flow Control Mechanism”, *Computers & Fluids*, 88: 678-687, (2013).

[12] Pirzadeh, S.Z. and Frink, N.T., “Assessment of the Unstructured Grid Software TetrUSS for Drag Prediction of the DLR-F4 Configuration”, *AIAA*, 2002-0839, (2002).

[13] Langtry R.B., Kuntz, M. and Menter, F., “Drag prediction of engine air frame interference effects with CFX-5”. *Journal of Aircraft*, 42(6): 1523-1529, (2005).

[14] Kroll N., Rossow, C.C., Schwamborn, D., Becker, K. And Heller, G., “MEGAFLOW—a numerical flow simulation tool for transport aircraft design”, *Proceedings of ICAS Congress*, 1105.1-1105.20, (2002).

[15] Schütte A., Einarsson, G., Madrane, A., Schöning, B., Mönnich, W. and Krüger, W.B., “Numerical simulation of maneuvering aircraft by CFD and flight mechanic coupling”, *RTO Symposium*. (2002).