UDK: 533.6.07:532:629.76
COSATI: 16-02, 20-04, 14-02

Numerical Investigation of NASA Tandem Control Missile and Experimental Comparison

Ali Akgül [1)]
Hayri Yiğit Akargün[1)]
Bora Atak[1)]
Abdullah Emre Çetiner[1)]
Ozan Göker[1)]

In this article, the aerodynamic performance of NASA Tandem Control Missile configuration is predicted using the FLUENT commercial CFD package and Engineering-level codes: Missile DATCOM, Aeroprediction, MISL3 and MISDL. The predictions of the engineering-level missile aerodynamic codes and the CFD results on the NASA Tandem-Control Missile are compared with wind tunnel data in order to evaluate the capabilities of these methods. The goal is to provide sample results to illustrate those flow regimes in which alternate prediction methods produce reasonable aerodynamic characteristics, and then to discuss some guidelines for the selection and use of alternate methods. Another purpose of these results is to illustrate those flow structures which require the use of advanced CFD calculations to ensure that accurate and valid results are obtained.

Key words: **experimental aerodynamics, projectile aerodynamics, tandem projectile, aerodynamic characteristics, aerodynamic testing, numerical methods, numerical fluid dynamics, wind tunnel.**

Notation and symbols

α – Angle of attack
M – Mach number
C_m Pitching moment coefficient
C_N Normal force coefficient
C_L Lift force coefficient
C_D Drag force coefficient

Introduction

NASA Tandem Control Missile (NTCM) is a supersonic wind tunnel test case configuration. The NTCM configuration is used in the "Applied Vehicle Technology (AVT) Panel Group 082 of the Research Technology Organization (RTO)" [1] for the validation of aerodynamic analysis tools. The test results presented in this paper are the wind tunnel test results of NASA Langley.

The primary aim of this study is to evaluate the capabilities of different aerodynamic analysis tools by comparing them with experimental data. As a result, a guideline for the selection and use of alternative aerodynamic calculation methods is to be obtained.

The second purpose of this study is to illustrate the flow regimes which require the use of advanced CFD calculations to ensure accurate and valid aerodynamic results.

The solutions in this paper were obtained using semi-empirical, engineering level codes; Missile DATCOM, MISL3, Aeroprediction (AP), MISDL and commercial CFD code FLUENT. Also, the results for different versions of Missile DATCOM, MISL3 and AP are presented in order to highlight differences between code releases.

Model

The NASA Tandem Control Missile was chosen as the main model for the Applied Vehicle Technology (AVT) Panel Group 082 of the Research Technology Organization (RTO). The aim of that study was to predict the flow field and performance characteristics of complex shaped projectiles at high Mach numbers and to investigate turbulence models.

A large-scale NASA Tandem Control Missile configuration is shown in Fig.1. The NTCM has the same canard and tail fin geometries in the x configuration and the body is a cylindrical body of revolution with an ogival nose.

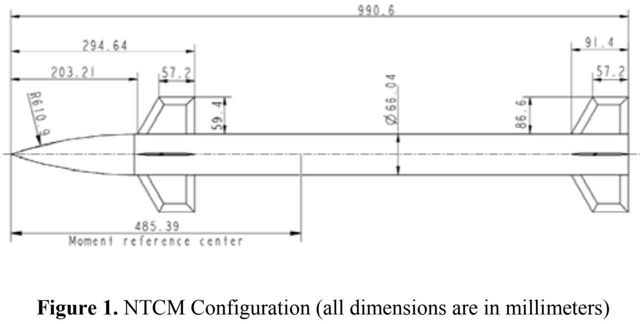

Figure 1. NTCM Configuration (all dimensions are in millimeters)

A small-scale NTCM model is also modeled in CFD computations in order to obtain a comparison for flow

[1)] ROKETSAN Missile Industries Inc., P.K.:30, 06780 Elmadağ-Ankara-TÜRKIYE visualization and numerical results of wind tunnel experiments. A scaled NTCM model in the wind tunnel test section is shown in Fig.2. The scale factor for this model is 0.263.

Figure 2. Small-scale NTCM Model [1]

Semi-empirical, engineering level codes

Engineering level codes are employed in order to estimate aerodynamic characteristics of missiles in conceptual and preliminary design phases for which fast calculation is of importance. In this paper, several versions of Missile DATCOM, MISL3 and AP codes are implemented.

Missile DATCOM: The Missile DATCOM is a semi-empirical code which is developed by the U.S. Air Force Flight Dynamics Laboratory for missile and rocket aerodynamic parameter estimation [2]. This software calculates static aerodynamic forces and moments and dynamic derivatives for missile configurations. The Missile DATCOM can be used to estimate stability and control characteristics in conceptual and preliminary design. ROKETSAN is an official user of the Missile DATCOM versions 5/97, 7/07, 8/08, 9/09.

The equivalent angle of attack and component build-up methods are implemented in the Missile DATCOM in addition to other semi-empirical methods for respective flight regimes. As a result one can obtain aerodynamic solutions for a wide range of conventional missile designs.

MISL3: The MISL3, which is developed by the Nielsen Engineering and Research Inc. (NEAR), uses an engineering level prediction method for aerodynamic performance prediction [3]. This software is based on the component build-up and the equivalent angle of attack method. In addition to this, the MISL3 incorporates Triservice fin-on body force and moment data base for a wide range of configurations and flow conditions. ROKETSAN is an official user of the MISL3 versions 2006, 2010, 2011.

The MISL3 has the capability to track vortices shed from the missile body and fins. The vortex modeling in the MISL3 enables the estimation of nonlinear aerodynamic forces and moments.

Aeroprediction (AP): Aeroprediction is a semi-empirical code that computes aerodynamics on most tactical weapons [4]. It uses analytical methods for low angles of attack flight and empirical databases at higher angles of attack to approximate nonlinear aerodynamics. Additionally, AP can perform trim based trajectory calculations. ROKETSAN is an official user of the AP versions 05, 09, 10, 11.

MISDL: The MISDL is a missile aerodynamic prediction program developed by the NEAR. The software is based on the panel method and employs intermediate-level methods to calculate aerodynamic force distributions, component loads, and overall longitudinal and lateral forces and moments for subsonic and supersonic flight [5].

The MISDL software can model circular and noncircular cross section bodies with fins having different planforms. The effects of rotational rates and nonuniform flow can be included in the aerodynamic analysis of the MISDL.

The methodology includes conformal mapping for noncircular body cross sections, and source/sink and doublet distributions to model the transformed axisymmetric body. Panel methods are applied to fin sections. The nonlinear effects of the body and fin wake vortices are included in the aerodynamic analysis [5]. ROKETSAN is an official user of the MISDL version 2011.

CFD simulation

Viscous CFD simulations are performed to calculate the flow-field and steady aerodynamic coefficients. The large-scale NTCM model is analyzed at Mach 1.75 between -4 and 28 degrees of the angle of attack. The analysis results are used to calculate aerodynamic drag, lift and pitching moment coefficients (C_D, C_L and C_m respectively). The calculated coefficients are compared with experimental results [1]. Also, the small-scale NTCM model is analyzed at Mach 2.0 and 24 degrees of the angle of attack. The result of this solution is used to visualize shock waves in the flow-field and it is compared with Leopold's [1] shadowgraph experiment.

Computational Grid

At the beginning of CFD studies, a grid convergence study is performed at Mach 1.75 and 10 degrees of the angle of attack for the large-scale NTCM model.

Five grids are generated for the grid convergence study. They are of different size but have the same type of grid elements. A cylindrical volume domain which has 30m in diameter and 45m in length is created for each grid. The distances of the computational domain surfaces are chosen far enough in order not to be affected by flow around the solid model. The computational domain grids for the CFD analysis are generated using GAMBIT and TGRID commercial programs. Triangular elements are generated to model the surface of the NTCM. For the boundary layer grid, 22 layers of prism cells are generated according to the chosen y^+ value about 1.0.

The post-process study showed that the y^+ value is in the range of 0.0-1.0 on the model. Fig.3 shows the y^+ values for the nose and body of the NTCM model at Mach 1.75 and 0 degree of the angle of attack.

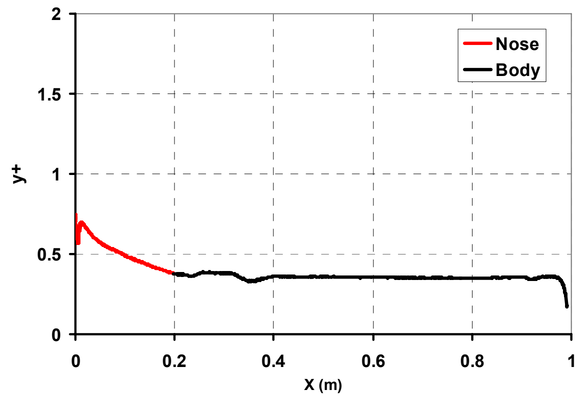

Figure 3. y^+ Value on Nose and Body

Finally, the computational domain is completed using unstructured tetrahedral elements. The total cell number of grids is shown in Table 1.

Table 1. Total Cell Number in Computational Domain

Coarsest Grid	2.20E+06 cells
Coarse Grid	4.34E+06 cells
Medium Grid	6.71E+06 cells
Fine Grid	9.31E+06 cells
Finest Grid	1.33E+07 cells

These grids are solved at Mach 1.75 and 10 degrees of the angle of attack. The normal force coefficient (C_N) and the pitching moment coefficient (C_m) are examined for the grid convergence. Fig.4 and 5 show the changes of C_N and C_m according to the number of cells in the computational domain.

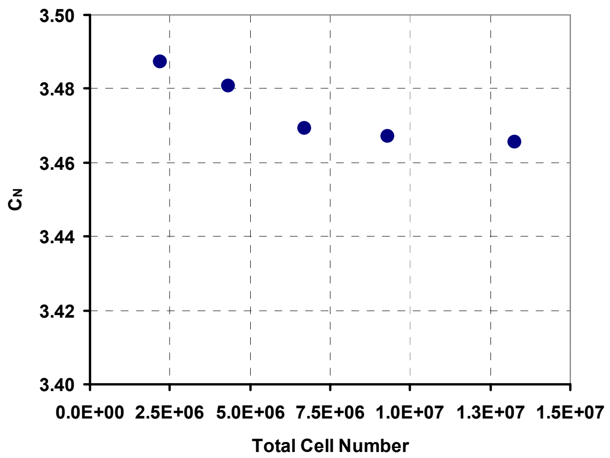

Figure 4. The Change of the Normal Force Coefficient According to the Total Cell Number

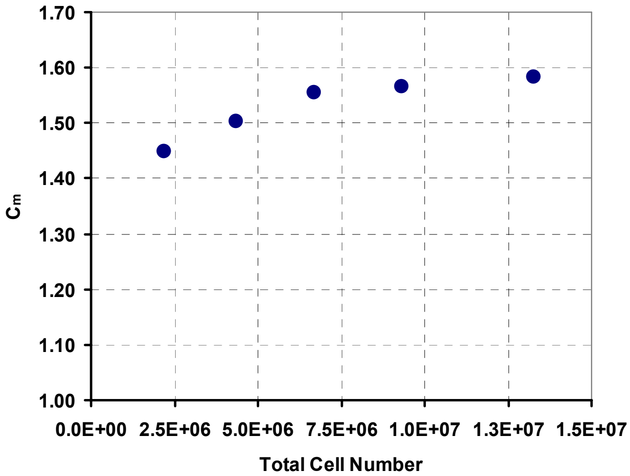

Figure 5. The Change of the Pitching Moment Coefficient According to the Total Cell Number

As it can be seen in Figs.4-5, C_N and C_m are converged at around the medium grid which has 6.7 million cells. For finer grids, the solution is slightly changed. Finer grids can be used only if the solution time has a similar trend with the changes of C_N and C_m. In Fig.6, the CPU times are shown for each solution.

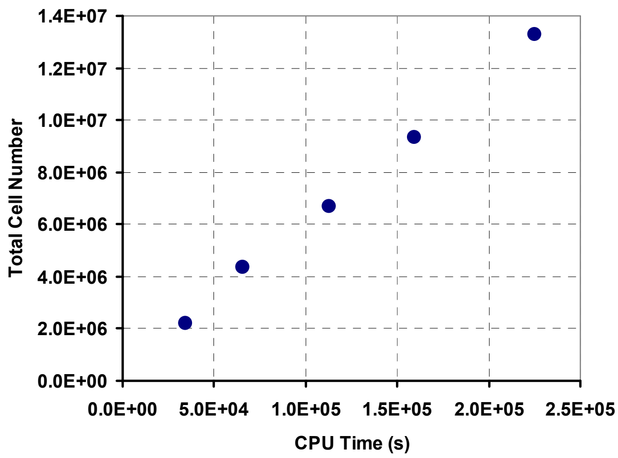

Figure 6. The Change of the CPU Time According to the Total Cell Number

The total cell number and the CPU time have a linear relation. Therefore, the usage of finer grids is not cost-effective. As a result of the grid convergence study, the medium grid which has 6.7 million cells is selected for CFD studies of the large-scale NTCM model. The surface grid and the cross section of the computational domain are shown in Fig.7 and 8.

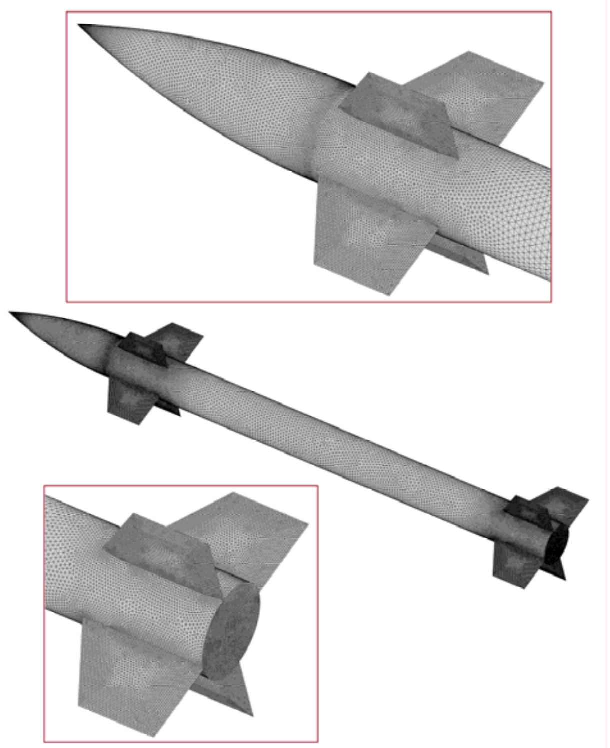

Figure 7. Surface Grid for NTCM Models

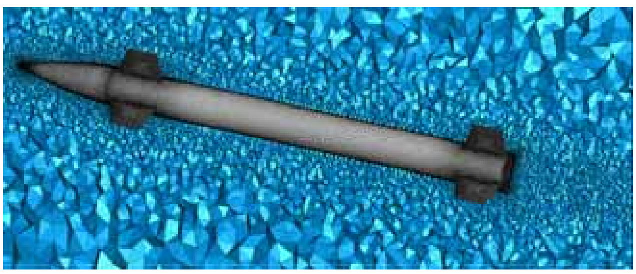

Figure 8. Cross Sectional View of the Computational Domain

A similar grid is generated for the small-scale NTCM model. Moreover, static pressure gradient adapted grid refinement is performed on this grid in order to get a better solution around shock waves. The adapted cells and the final grid after adaption are shown in Fig.9.

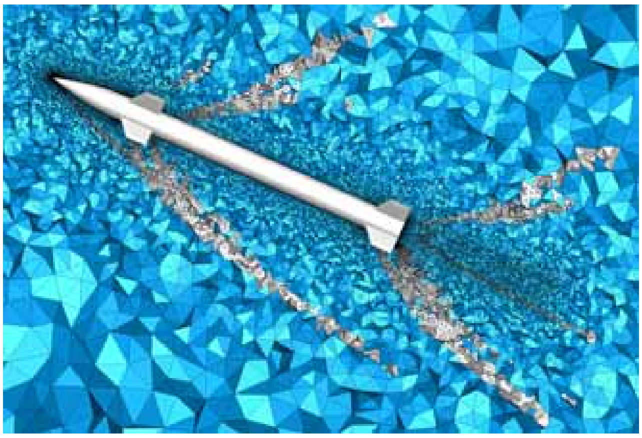

Figure 9. Adapted Grid for the Small-scale NTCM Model (Adapted cells are marked with white)

Flow Solver and Boundary Conditions

Three dimensional, viscous, compressible, steady Navier-Stokes Equations are solved by using the FLUENT commercial flow solver. Steady, density based solver option is used with *k-ε*, *k-ω* and Spalart Allmaras turbulence models [6]. A comparison study is performed to select the most suitable turbulence model. Conservation equations are solved by the finite volume technique.

$$\frac{\partial}{\partial t}\int_V W dV + \oint [F - G]\cdot dA = \int_V H dV \quad (1)$$

$$W = \begin{Bmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ \rho E \end{Bmatrix}, F = \begin{Bmatrix} \rho v \\ \rho v u + p i \\ \rho v v + p j \\ \rho v w + p k \\ \rho v E + p v \end{Bmatrix}, G = \begin{Bmatrix} 0 \\ \tau_{xi} \\ \tau_{yi} \\ \tau_{zi} \\ \tau_{ij} v_j + q \end{Bmatrix} \quad (2)$$

The inviscid flux vector F is evaluated by a standard upwind flux-difference splitting. In the density based solver, each equation in the coupled set of governing equations is linearized implicitly with respect to all dependent variables in the set, resulting in a block system of equations. A block Gauss-Seidel, point implicit linear equation solver, is used with an algebraic multigrid method to solve the resultant block system of equations. Second-order discretization was used for all flow variables.

Outer boundaries of the computational domain are set as far-field, with sea-level temperature and pressure free stream conditions (300 K, 101325 Pa). All the solid surfaces are modeled as no-slip, adiabatic wall boundary conditions.

For the calculation of the aerodynamic coefficients, the dimensions of the large-scale NTCM model are used. The model reference length is 0.06604 m which is the diameter of the body, and the moment reference point is 0.48539 m aft of the model nose. The reference area is taken as 0.00343 m^2 which is the cross-sectional area of the body.

The large-scale NTCM model's grid is solved with k-ε, k-ω and Spalart Allmaras turbulence models at Mach 1.75 between 0 and 24 degrees of the angle of attack and the results are compared with experimental data. In Figs.10-12, the comparisons of the drag coefficient, lift coefficient and moment coefficient are shown respectively.

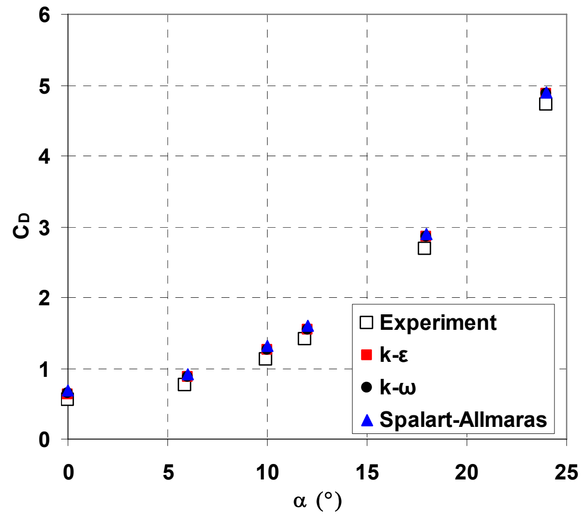

Figure 10. Drag Coefficient Comparison

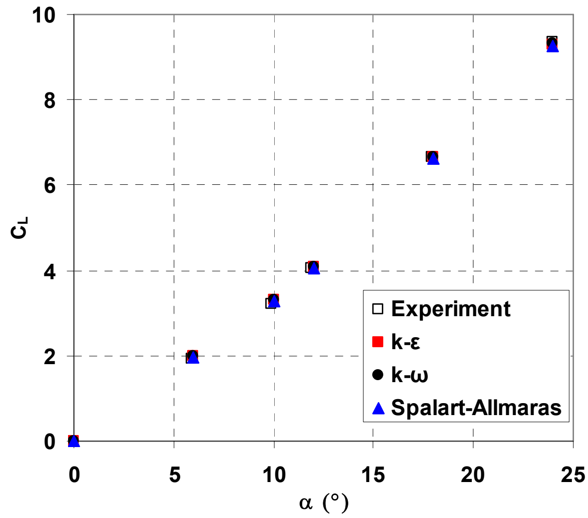

Figure 11: Lift Coefficient Comparison

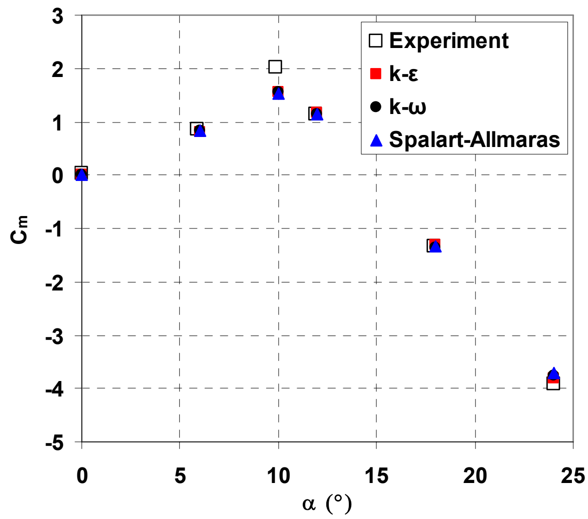

Figure 12. Moment Coefficient Comparison

The results of all three different turbulence models compare well with experimental data. However, the Spalart Allmaras turbulence model is slightly different from the experimental result at high angles of attack. In order to select the turbulence model that is used, the CPU times of the converged solutions at 10º angles of attack are compared in Table 2.

Table 2. Solution Times

Turbulence Model	CPU Time (s)
k-ε	23700
Spalart Allmaras	22069
SST-k-ω	68695

The Spalart Allmaras and k-ε turbulence models are much faster than the k-ω turbulence model. When the slight difference in the pitching moment at high angles of attack is considered, k-ε is selected for the rest of the study.

Computational Grid

There are some examples of the flow-field post-proccess studies of the large-scale NTCM model shown in Figs.13-15.

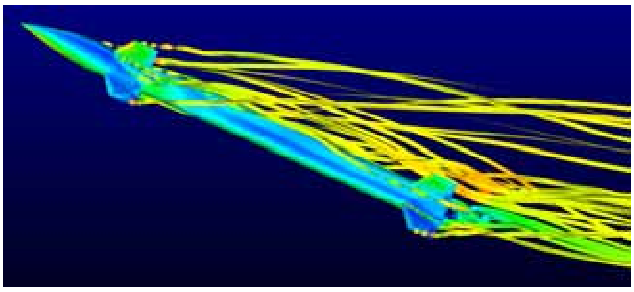

Figure 13. Pressure Contours and Streamlines at M=1.75, α=24°

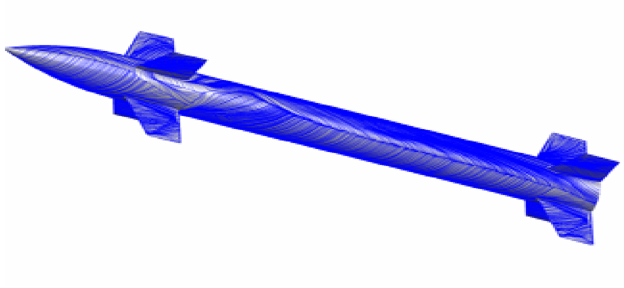

Figure 14. Surface Streamline Pattern at M=1.75, α=24°

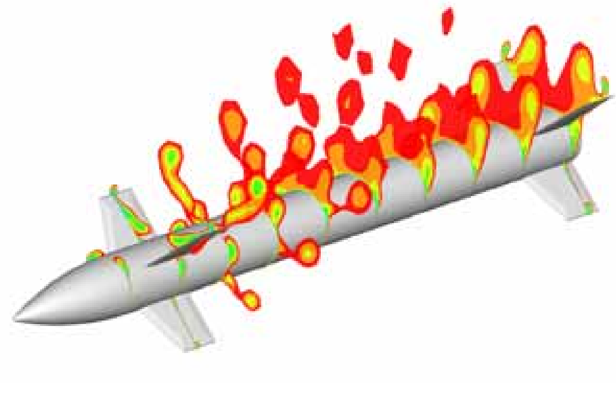

Figure 15. Vortices Visualization at M=1.75, α=24°

Fig.16 shows the shadowgraph experiment result which shows shock waves on the small-scale NTCM model at Mach 2.0 and 24 degrees of the angle of attack [1]. Also, a shock wave visualization study is performed using the CFD solution of the small-scale NTCM model at the same condition. As seen in Fig.17, the oblique shock waves on the nose, canard and body sections are well captured by the pressure adapted grid of the small-scale NTCM model.

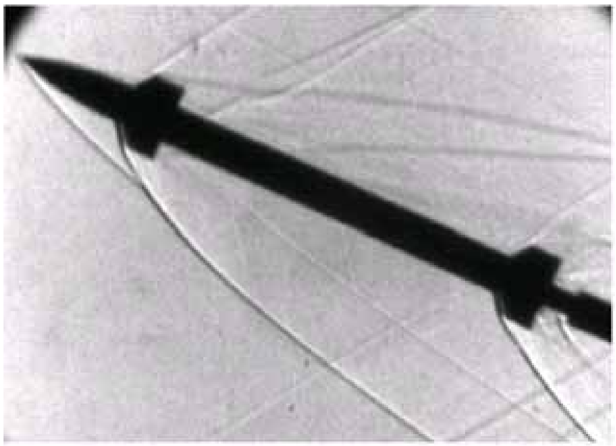

Figure 16. Shadowgraph Experiment, M=2.0, α=24° [1]

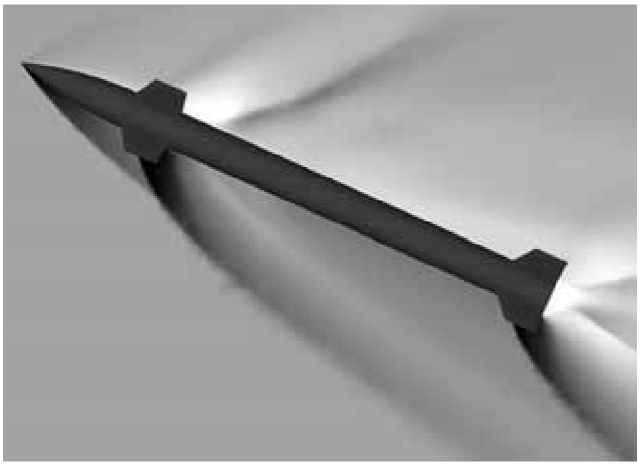

Figure 17. Shock Wave Visualization, M=2.0, α=24°

Experimental data

The NTCM experimental data is obtained from the wind tunnel tests performed at NASA Langley [1]. The test conditions, which are used for comparisons, are as follows:

Mach: 1.75

Angle of Attack: $-4° < \alpha < 28°$

Reynolds Number: 6.6×10^4 /cm

The wind tunnel tests with non-zero control deflections were also performed at NASA Langley. Since these test conditions were not incorporated in AVT Panel Group 082 of RTO, the wind tunnel test results with non-zero control deflections are not available and not included in this study.

Prediction results and comparison

In this section the results obtained by semi-empirical, engineering level codes and CFD tools are presented in comparison with the experimental results. The reference length for the aerodynamic coefficient is the body diameter.

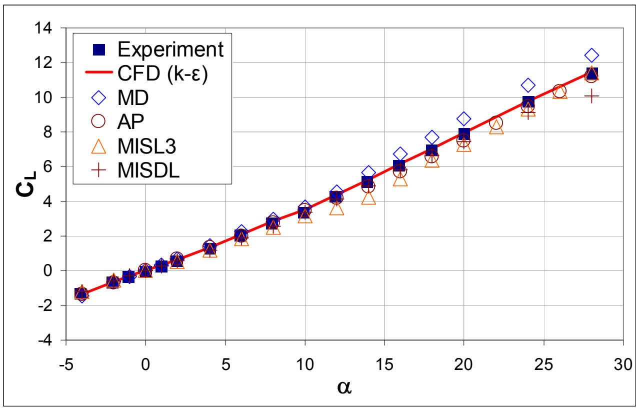

Figure 18. Comparison of the experimental lift force coefficient with the CFD and the semi-empirical results at M=1.75

The lift force coefficient results of experiment, CFD and engineering level codes at Mach 1.75 are shown in Fig. 18. The CFD results can be considered to be exactly the same as the experimental data for all angles of attack. The engineering level codes are also in good agreement with the experiment.

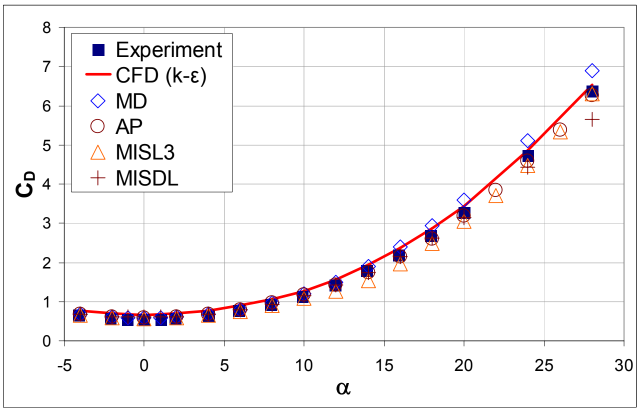

Figure 19. Comparison of the experimental drag force coefficient with the CFD and the semi-empirical results at M=1.75

The drag coefficient results of experiment, CFD and engineering level codes at Mach 1.75 are shown in Fig.19. The CFD and the semi-empirical codes results compare well with the experimental data.

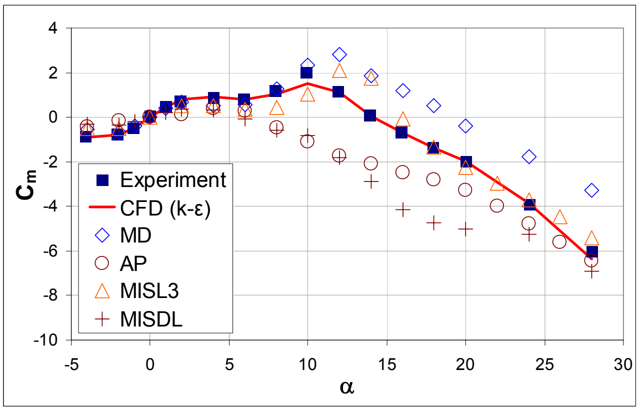

Figure 20. Comparison of the experimental pitching moment coefficient with the CFD and the semi-empirical results at M=1.75

The pitching moment coefficient results of experiment, CFD and engineering level codes at Mach 1.75 are shown in Fig.20. The CFD computations predict the experimental data accurately for the whole angle of attack range. The MISL3 gives more accurate results compared to other semi-empirical codes for most of the angles of attack. The engineering level codes other than the MISL3, predict poorly at the angles of attack wider than 8°.

Conclusion

In this paper, the numerical investigation of the NTCM configuration with CFD and semi-empirical codes is performed and aerodynamic coefficients are calculated. The details of the CFD simulation are presented as a grid structure, surface streamline pattern, vortex structure and shock wave visualization. The calculated aerodynamic coefficients are compared with the experimental data of the wind tunnel tests which are available in the scope of "AVT Panel Group 082 of the RTO". Upon this comparison, it can be seen that the CFD results are in the best agreement with the experimental data. In the calculation of lift and drag force coefficients, all engineering-level codes give good results. For the pitch moment calculation, the MISL3 results are more accurate when compared to other engineering-level codes.

In this article, different modeling techniques are implemented for numerical methods. The methodology and solution strategy are validated in order to use the same techniques in the solution of similar problems or cases.

Acknowledgment

The authors thank ROKETSAN for supporting this research.

References

[1] MENDENHALL,M.R., LEAVITT,L.D., LEOPOLD,F.: *Experimental test cases*, Assessment of Turbulence Modeling for High-Speed Vehicles, RTO Applied Vehicle Technology Panel (AVT) TG-082, April 2005.

[2] BLAKE,W., AUMAN,L., DOYLE,J., ROSEMA,C., UNDERWOOD,M.: *Missile Datcom User's Manual*, Air Research Laboratory, 2008.

[3] LESIEUTRE,D.J.: *MISL3 Aerodynamic Analysis For Finned Vehicles With Axisymmetric Bodies*, Nielsen Engineering and Research, 2010.

[4] Moore,F.G.: *AP09 Theory And User's Course*, Aeroprediction, Inc., 2009.

[5] LESIEUTRE,D.J.: *Detailed Aerodynamic Prediction Program MISDL Code User's Manual*, Nielsen Engineering and Research, 2010.

[6] Fluent 12.0.16 Users Guide, Fluent, Inc.

Received: 28.11.2011.

Poređenje numeričkih i eksperimentalnih rezultata projektila sa NASA tandem projektilom

U ovom radu su izvršena poređenja aerodinamičkih performansi projektila sa NASA tandem kontrolnim projektilom koristeći komercijalni FLUENT CFD softverski paket i raspolološivih poluempirijskih metoda koje su implementirane u pakete: Missile DATCOM, Aeroprediction, MISL3 i MISDL. Procene poluempirijskih metoda i CFD rezultata projektila sa NASA tandem kontrolnim projektilom su upoređena sa dostupnim eksperimentalnim rezultatima kako bi se sagledao kvalitet proračunskih metoda. Cilj istraživanja je da se ilustruje mogućnost i tačnost primene alternativnih poluempirijskih metoda za sračunavanje aerodinamičkih karakteristika, a potom se daju određene preporuke za izbor i korišćenje alternativnih metoda. Druga svrha ovih rezultata je da se ilustruje koje konstrukcije zahtevaju korišćenje savremenih CFD mumeričkih metoda za opstrujavanje kojima se obezbeđuje potrebna tačnost i valjanost dobijenih rezultata

Ključne reči: **eksperimentalna aerodinamika, aerodinamika projektila, tandem projektil, aerodinamičke karakteristike, aerodinamičko ispitivanje, numeričke metode, numerička dinamika fluida, aerodinamički tunel.**

Сравнение численных и экспериментальных результатов снарядов с тандемной NASA-ракетой

В этой работе выполнены сравнения аэродинамических характеристик снарядов с тандемной NASA-ракетой контрольного управления с использованием коммерческого программного обеспечения пакета FLUENT CFD и полуэмпирических доступных методов, реализованных в пакете: ракеты DATCOM, Aeroprediction, MISL3 и MISDL. Оценки полуэмпирических методов и результатов CFD снарядов с тандемной NASA-ракетой контрольного управления сопоставлены с доступными экспериментальными результатами для того, чтобы реализовать качество аналитических методов. Основной целью является демонстрация возможностей и точности применения альтернативных полуэмпирических методов для расчёта аэродинамических характеристик, а затем надо дать определённые рекомендации по выбору и использованию альтернативных методов. Другая цель этих результатов, чтобы показать, какие структуры требуют использования современных CFD численных методов для обтекания, необходимых для обеспечения точности и достоверности полученных результатов.

Ключевые слова: **экспериментальная аэродинамика, ракетная аэродинамика, тандемные снаряды, аэродинамические характеристики, аэродинамические испытания, обтекание, программное обеспечение, численные методы, вычислительная гидродинамика(CFD) , аэродинамическая труба.**

Comparaison des résultats numériques et expérimentaux chez les missiles avec le missile tandem NASA

Dans ce travail on a comparé les performances aérodynamiques des missiles avec le missile tandem de contrôle NASA en utilisant le progiciel commercial FLUENT CFD et les méthodes semi empiriques disponibles qui font partie des progiciels suivants: Missile DATCOM, Aeroprediction, MISL3 et MISDL. Les estimations des méthodes semi empiriques et des résultats CFD des missiles avec le missile tandem de contrôle NASA ont été comparés avec les résultats expérimentaux pour évaluer la qualité des méthodes citées. Le but de ces recherches est d'illustrer la possibilité et la précision d'emploi des méthodes semi empiriques alternatives pour le calcul des caractéristiques aérodynamiques et de donner les recommandations pour le choix et l'utilisation des méthodes alternatives . L'autre but des résultats obtenus est de présenter les constructions qui exigent l'application des méthodes numériques CFD pour les courants par lesquels on assure la précision et la validité des résultats réalisés.

Mots clés: **aérodynamique expérimentale, aérodynamique de missile, missile tandem, caractéristiques aérodynamiques, essai aérodynamique, courant, logiciel, méthodes numériques, dynamique des fluides d'ordinateur, tunnel aérodynamique.**