UDK: 620.178:623.463.5
COSATI: 20-04, 16-02, 16-04

Numerical Simulations of Store SeparationTrajectories Using the EGLIN Test

Yunus Emre Sunay[1)]
Emrah Gülay[1)]
Ali Akgül[1)]

One of the most important topics for missile development projects is safe separation of the missile from air vehicles. Store separation tests are expensive, time consuming and dangerous since tests can end up with fatal accidents; therefore, safe separation simulations and missile trajectory predictions by using numerical methods became a critical issue. In this study, the EGLIN test case is used to validate the predicted trajectory of a store separated from a wing. The EGLIN test case is a generic store separation test case and most commonly used in the validation of trajectory prediction tools. the experiments were performed in the transonic (M=0.95) and supersonic (M=1.2) flow regimes. The EGLIN store separation test case is simulated by using the time-dependent Computational Fluid Dynamic (CFD) analysis. Coupled six degree of freedom (6DOF) and flow solvers are used to predict the store trajectory. In addition, Navier-Stokes and Euler computations are performed to investigate the viscous effects on the store trajectory analysis.

Key words: **computational fluid dynamics, numerical simulation, Navier-Stokes equations, air missile, store trajectory, store separation, EGLIN test.**

Notation and symbols

Φ – Roll angle
Θ – Pitch angle
Ψ – Yaw angle
C_p – Pressure coefficient

Introduction

SAFE separation of operational or newly designed missiles from air vehicles is a critical issue in terms of missile integration process. Detailed engineering analyses, wind tunnel and flight tests are needed to be performed.

In the view of recent developments in computational methods, the numerical analysis can replace flight and wind tunnel tests in some cases for certification processes. Even validated computational methods can be used to complete a safe separation process. This will result in a cost and time effective integration study. In this paper, the EGLIN test case is used to validate an engineering approach to simulate store separation at transonic and supersonic speeds [1,4]. The analysis results are compared with the experimental values.

CFD Modeling & Simulation

In this part, CFD modeling and the simulation of an EGLIN test case model will be explained. GAMBIT (v2.4.6) and TGRID (v5.0.6) are used to generate computational grids for a CFD analysis. Also, the FLUENT commercial program (v12.0.16) is used as a solver. The sign convention, the test case model, the computational grids and the results of the analysis will be explained in detail in the following sections [8,9].

Test Model Sign Convention

A generic EGLIN test model is used in the validation of the CFD analysis. The test was conducted at the Arnold Engineering Development Center in the Aerodynamic Wind Tunnel (4T) in 1990. The EGLIN test model has three parts; wing, pylon and finned body. The sketch of the EGLIN test case model and the sting used in the wind tunnel test are represented in Fig.1 [5].

The wing consisted of a clipped delta wing with 45° sweep and a constant NACA 64A010 airfoil section. The pylon has an ogive-flat plate-ogive cross section shape. The store body was an ogive-cylinder-ogive with an aft cylindrical sting. The fins on the store consisted of a clipped delta wing with a 45° sweep and a constant NACA 0008 airfoil section. The gap between the pylon and the finned body is 0.1778 cm. The geometric specifications of the body are represented in Fig.2.

1) ROKETSAN Missile Industries Inc., P.K.30: 06780 Elmadağ-Ankara,TŰRIKYE

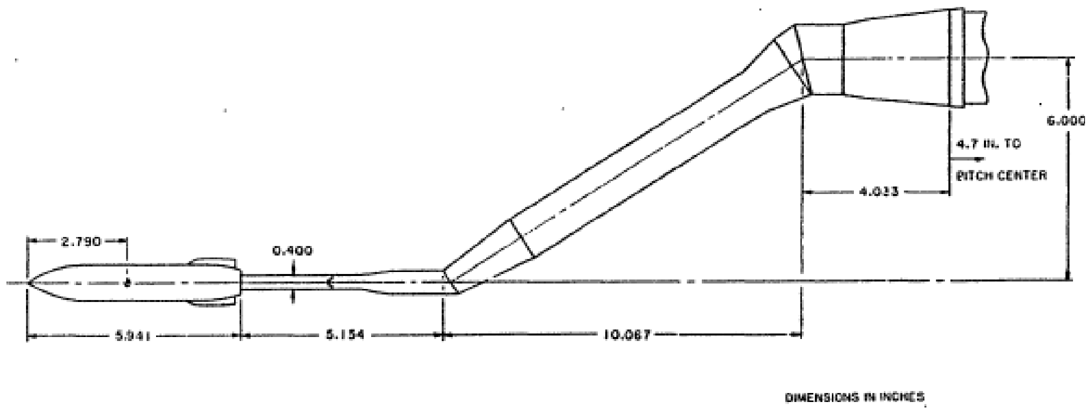

Figure 1. Sketch of the EGLIN test case

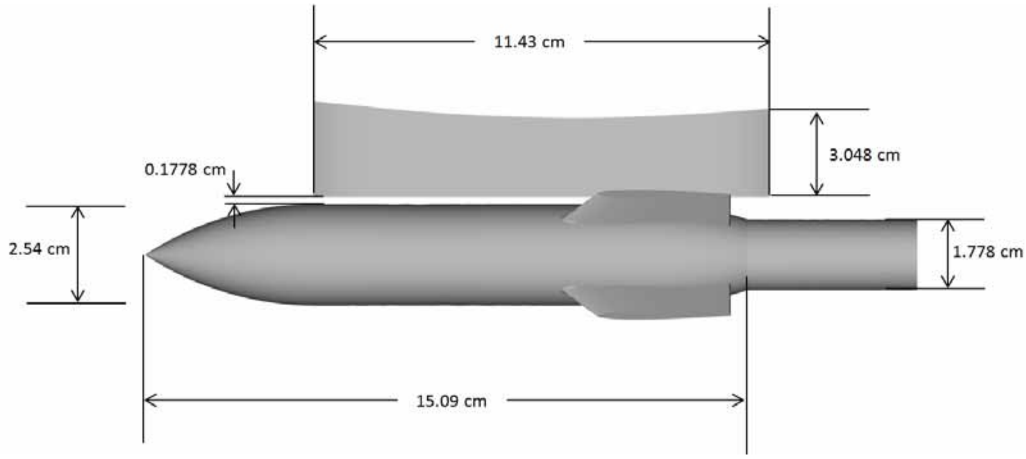

Figure 2. Geometric specifications of the EGLIN test case

Experimental Data

The experiments were conducted in transonic (M=0.95) and supersonic (M=1.2) flow regimes. The position and the orientation of the store are obtained during the test for both flow regimes. The surface pressure distribution on the model is only available for the transonic flow regime.

The sign convention used for the calculation of the computational fluid dynamics simulations and experiments is given in Fig.3.

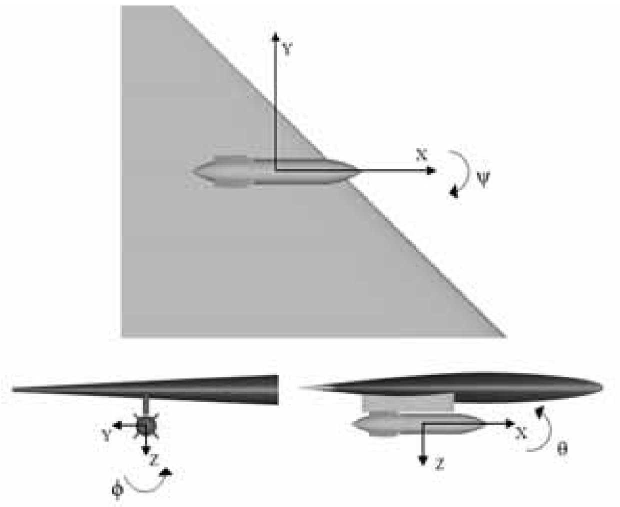

Figure 3. Model and sign convention

In the test, aft and forward ejector forces are applied to the store to provide safe separation. The store inertial/mass parameters and ejector parameters are given in Table 1 [3].

Table 1. Store inertial/mass and ejector parameters

Mass	907 kg
Center of Mass	1417mm (aft of store nose)
Roll moment of inertial	27 kg.m^2
Pitch moment of inertial	488 kg.m^2
Yaw moment of inertial	488 kg.m^2
Forward ejector location	1237.5mm (aft of store nose)
Aft ejector location	1746.5mm (aft of store nose)
Forward ejector force	10.7kN
Aft ejector force	42.7kN

Solid Model & Computational Mesh

The EGLIN geometry was generated for CFD studies based on a test model used in the Arnold Engineering Development Center in the Aerodynamic Wind Tunnel. The geometry consists of three different parts; wing, pylon and store. The generated solid models are shown in Fig.4.

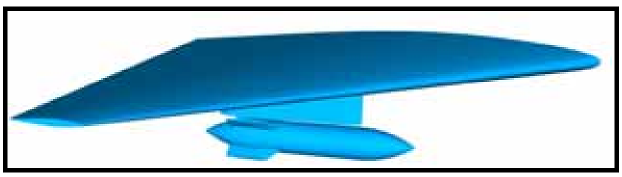

Figure 4. Solid model of the EGLIN test case.

The initially generated grid for Euler computations has 2,112,822 cells. Deformed computational grids at different time steps are given in Fig.5.

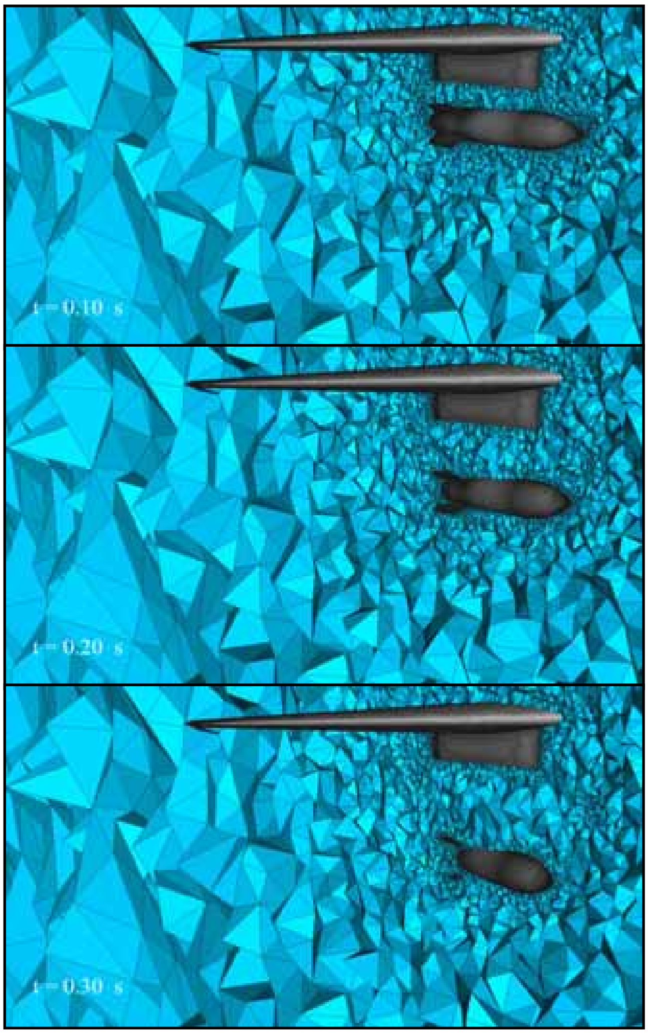

Figure 5. Computational grids for the CFD analysis using Euler equations at different time steps

The initial generated grid for the Navier-Stokes analysis has 3,412,792 cells and is shown in Fig.6.

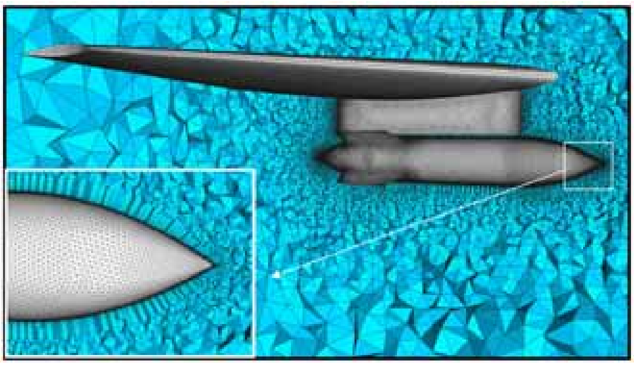

Figure 6. Computational grid for the CFD analysis using the NS equation (t=0sec)

The deformed computational grid for the Navier-Stokes analysis at the 0.2 seconds of the analysis is given in Fig.7. The boundary layer part of the computational grid for the Navier-Stokes analysis is not deformed during the solution.

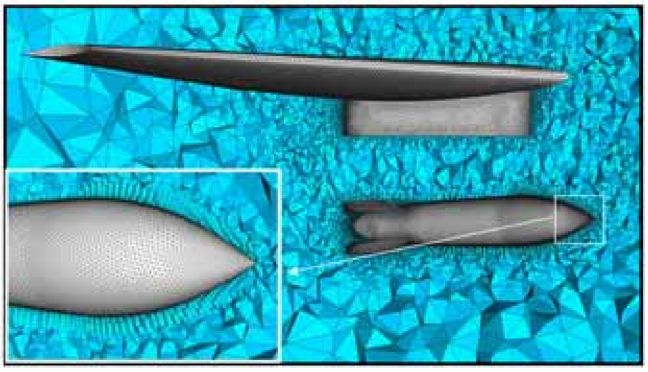

Figure 7. Computational grid for the CFD analysis using the NS equation (*t*=0.2sec)

The computational domain inlet was located at 17 wing length, outlet was located at 25 wing length, upper boundary was located at 17 wing length, lower boundary was located at 25 wing length and side boundary was located at 17 model length away from the center of the store. The solution domain is shown in Fig.8.

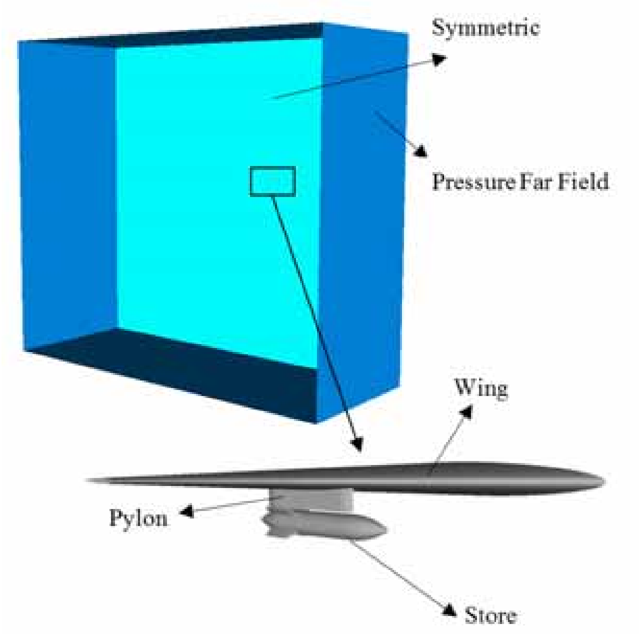

Figure 8. Solution domain and a part of the EGLIN test model

Flow Solver and Boundary Condition

The FLUENT commercial flow solver was used to predict the trajectory of the EGLIN test model by using Euler and Navier-Strokes Equations.

Euler

The implicit, compressible, unstructured-mesh solver was used. The three-dimensional, Euler equations were solved using the finite volume method [6,7]:

$$\frac{\partial}{\partial t}\int_V W dV + \oint F \cdot dA = \int_V H dV \qquad (1)$$

where

$$W=\begin{Bmatrix}\rho\\ \rho u\\ \rho v\\ \rho w\\ \rho E\end{Bmatrix}, F=\begin{Bmatrix}\rho v\\ \rho vu+pi\\ \rho vv+pj\\ \rho vw+pk\\ \rho vE+pv\end{Bmatrix} \quad (2)$$

The calculations took about 3 seconds of the CPU time per iteration and convergence was achieved in about 1,800 iterations for the steady part of the solution [10].

Navier-Stokes Equation

The three-dimensional, Reynolds-Average Navier-Stokes (RANS) equations were solved using the finite volume method [6,7]:

$$\frac{\partial}{\partial t}\int_V W dV + \oint [F-G]\cdot dA = \int_V H dV \quad (3)$$

where

$$W=\begin{Bmatrix}\rho\\ \rho u\\ \rho v\\ \rho w\\ \rho E\end{Bmatrix}, F=\begin{Bmatrix}\rho v\\ \rho vu+pi\\ \rho vv+pj\\ \rho vw+pk\\ \rho vE+pv\end{Bmatrix}, G=\begin{Bmatrix}0\\ \tau_{xi}\\ \tau_{yi}\\ \tau_{zi}\\ \tau_{ij}v_j+q\end{Bmatrix} \quad (4)$$

The boundary conditions are represented in Fig.8. Downstream, upstream, and all-side boundaries, except the right-side one, were set as far field (characteristics-based inflow/outflow), with a standard atmosphere model for 26,000 ft altitude temperature and pressure free stream conditions. The right-side boundary was defined as symmetric. Solid surfaces were modeled as no-slip, adiabatic wall boundary conditions for the Navier-Stokes analysis [3, 10].

The calculations took about 9 s of the CPU time per iteration and convergence was achieved in about 1,800 iterations for the steady part of the solution. The time step is 0.001 sec and 20 iterations were done for each time step.

In this simulation, Fluent uses an implicit, node-based finite volume scheme. Roe's flux difference splitting scheme is used to compute inviscid fluxes at the boundary of each control volume for the Navier-Stokes analysis and viscid fluxes at the boundary of each control volume for the Euler analysis. A second-order accurate, upwind extrapolation is used to determine the values of the flow variables at the boundary. The *k-ε* model was designed especially for aerospace applications involving wall-bounded flows and has been shown to give good results for boundary layers subjected to adverse pressure gradients. Turbulence modeling is achieved by using the wall treatment *k-ε* two-equation turbulence model which is suitable for y^+=1. The first height of the boundary layer of the computational grid was determined according to y^+=1. The y^+ values on the wing-pylon and the store are given in Fig.9.

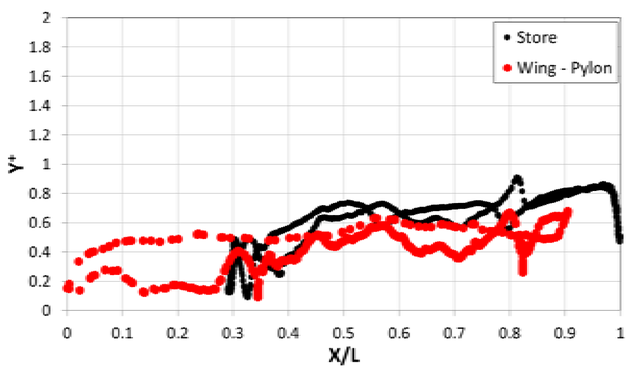

Figure 9. y^+ on the wing-pylon and the store

Convergence was determined by tracking the change in the flow residuals and the aerodynamic coefficients during the solution. The solution was deemed converged when the aerodynamic coefficients with respect to the time step had the same frequency and wave length in the solution time. Fig.10 shows residual changes with respect to iteration for the unsteady solution for the Navier-Stokes analysis.

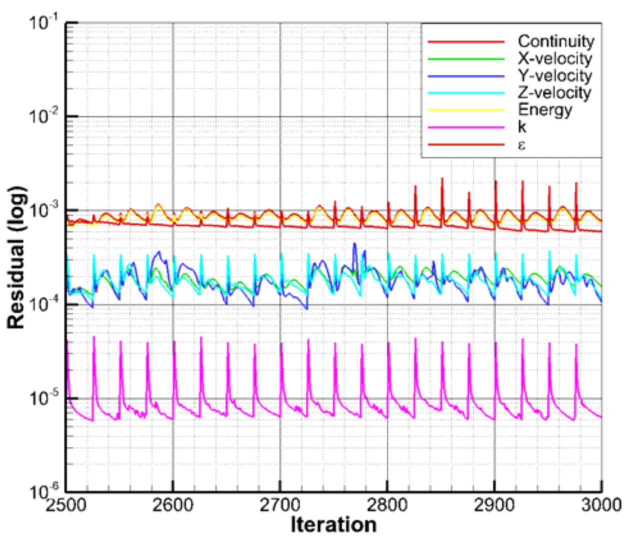

Figure 10. Residual versus iteration for the unsteady solution

Mesh Deformation Method

As the store starts to move due to gravity and aerodynamic forces acting on the body, the position of the store has to be changed on the mesh. If the displacements of nodes are large compared to the local cell sizes, cells can become degenerated. This will invalidate the mesh (e.g., result in negative cell volumes) and, consequently, lead to convergence problems. By checking the mesh quality, degenerated cells have to be smoothed or new cells have to be generated. Up to some quality criteria, skew cells are smoothed by the spring analogy method. If the quality of cells exceeds the predefined limit, the mesh has to be updated. Bad quality cells are replaced by created new cells using the re-meshing method. These two methods are explained briefly in the following sections [6,7].

Spring Analogy

In the spring-based smoothing method, the edges between any two mesh nodes are idealized as a network of interconnected springs. The initial spacing of the edges before any boundary motion constitutes the equilibrium state of the mesh. A displacement at a given boundary node will generate a force proportional to the displacement along all the springs connected to the node. Using Hook's Law, the force on a mesh node can be written as

$$\vec{F}_i=\sum_{j}^{n_i} k_{ij}(\Delta\vec{x}_i-\Delta\vec{x}_j) \quad (5)$$

where $\Delta\vec{x}_i$ and $\Delta\vec{x}_j$ are the displacements of the node i and its neighbor node *j*, n_i is the number of neighboring nodes connected to the node *i*, and k_{ij} is the spring constant between the node *i* and its neighbor *j*. The spring constant for the edge connecting the nodes i and j is defined as

$$k_{ij}=\frac{1}{\sqrt{|\vec{x}_i-\vec{x}_j|}} \quad (6)$$

At equilibrium, the net force on a node due to all the springs connected to the node must be zero. This condition results in an iterative equation such that

$$\Delta\vec{x}_i^{m+1} = \frac{\Sigma_j^{n_i} k_{ij} \Delta\vec{x}_j^{m}}{\Sigma_j^{n_i} k_{ij}} \qquad (7)$$

Since displacements are known at the boundaries (after the boundary node positions have been updated), Eq.7 is solved using a Jacobi sweep on all interior nodes. At convergence, the positions are updated such that

$$\vec{x}_i^{n+1} = \vec{x}_i^{n} + \Delta\vec{x}_i^{m,converged} \qquad (8)$$

where n+1 and n are used to denote the positions at the next time step and the current time step, respectively [6-7].

Re-meshing

In the re-meshing method, cells are marked based on cell skewness, minimum and maximum length scales as well as an optional sizing function. Each cell is evaluated and is marked for re-meshing if it meets one or more of the predefined criteria. If the cell skewness is greater than a specified maximum skewness, or the cell length scale is smaller than a specified minimum length scale, or the cell size is larger than a specified maximum length scale, marked cells are deleted and new cells are created [6,7].

Results

The experimental and numerical results are compared and presented in this part of the report.

Transonic Flow

Time-dependent CFD analyses were performed for Mach number 0.95 during 0.45 seconds. The experimental store position and orientation data are compared with the CFD analyses results in Fig.11 and 12, respectively.

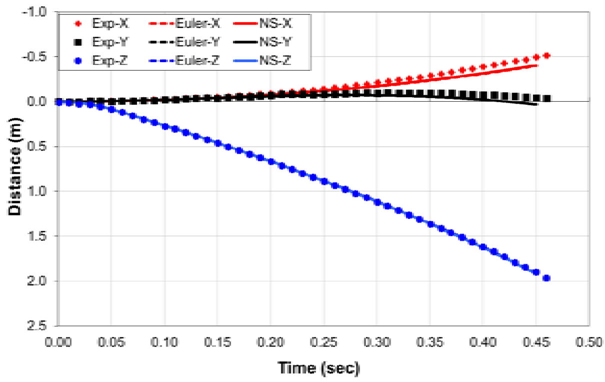

Figure 11. Trajectory of the center of gravity locations

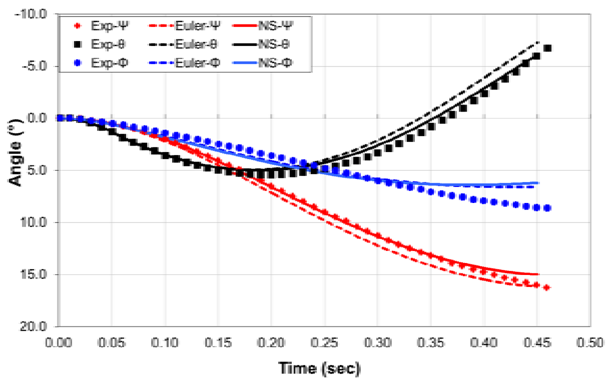

Figure 12. Trajectory of the angular orientation

The pressure distribution along the store body at the cross section by ϕ=5° with vertical axes for different time steps is shown in Figs. 13-15.

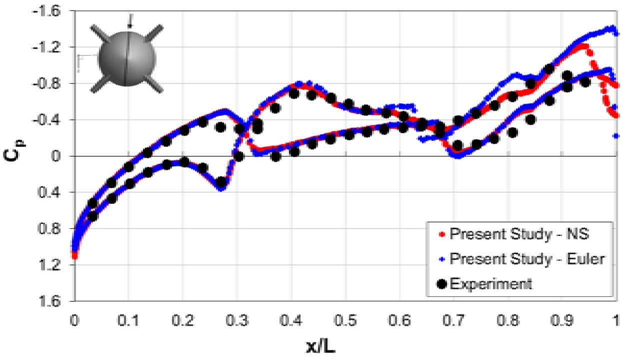

Figure 13. Pressure distribution of the store (t=0sec)

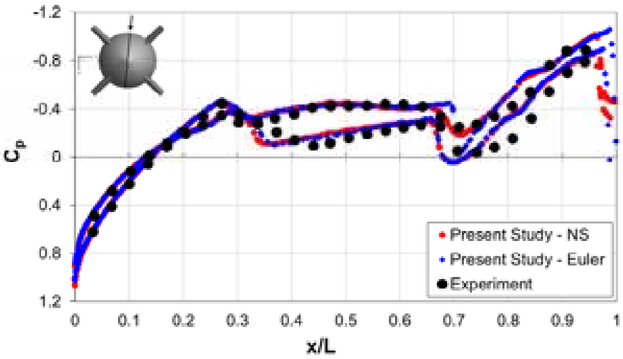

Figure 14. Pressure distribution of the store (t=0.17sec)

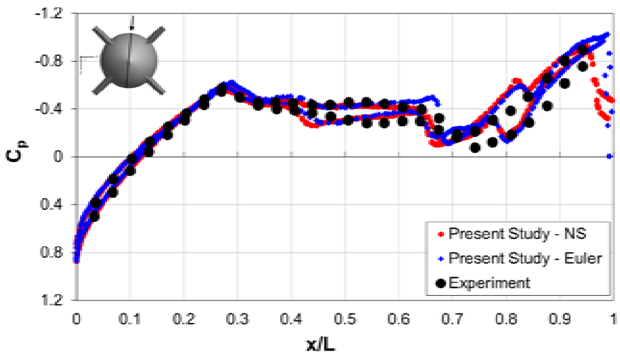

Figure 15. Pressure distribution of the store (t=0.32sec)

The experimental and numerical store position and orientation values are compared in Fig.16.

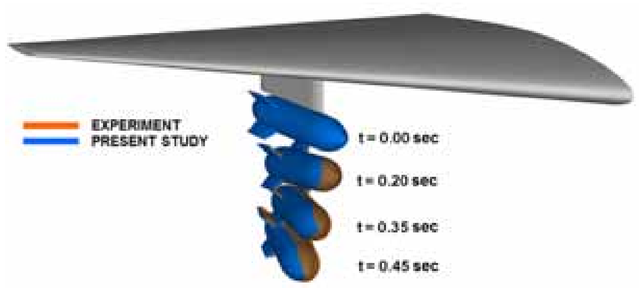

Figure 16. Position of the store w.r.t. time (M=0.95)

Supersonic Flow

The time-dependent CFD analyses for the supersonic store separation test case at Mach number 1.2 were performed for 0.8 seconds. The experimental store position and orientation data are compared with the CFD analyses results in Fig.17 and 18, respectively.

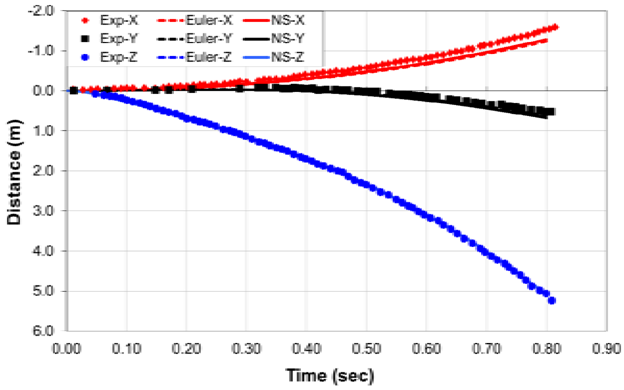

Figure 17. Trajectory of the center of gravity locations

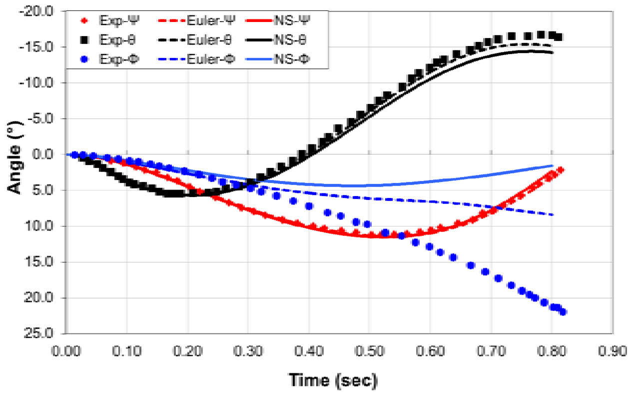

Figure 18. Trajectory of the angular orientation

The experimental and numerical store position and orientation values are compared in Fig.19.

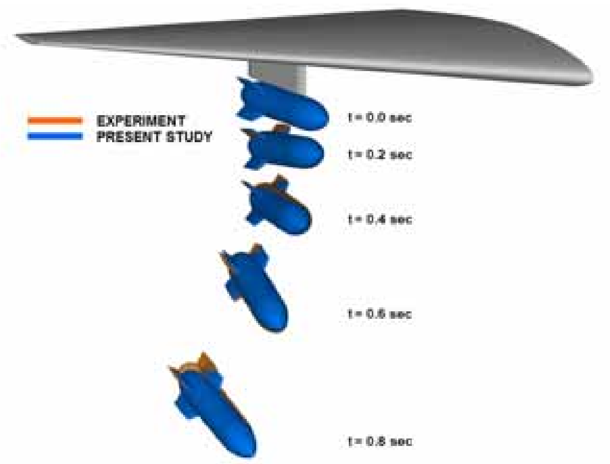

Figure 19. Position of the store w.r.t. time (M=1.2)

Conclusions

In this study, an engineering approach for store separation is studied by using a well-known generic store separation EGLIN test case. The store trajectory and orientation are predicted by using coupled 6DOF and Euler/Navier-Stokes equations. Grid deformation techniques and re-meshing algorithms are used to obtain a grid for the next time step. The Euler and Navier-Stokes results are compared with the experimental data and a good agreement is obtained. It can be concluded that the viscous effects have negligible influence on the results of this type of problems. Hence, the presented engineering approach can be used in the trajectory prediction of real store separation problems.

Acknowledgments

The authors thank ROKETSAN from Ankara for their support in the study of the problem of missile separation from the aircraft wing.

References

[1] PARIKH,P., PIRZADEH,S.: *Unstructured Grid Solution to a Wing/Pylon/Store Configuration*, Journal of Aircraft, 31(6) (1994)

[2] LEE,J.M.: *Studies of Combined Use of CFD and Wind Tunnel Test Approaches to Simulate a Store Separation form F-15E Using Efficient CFD Database Generation*, 22nd Applied Aerodynamics Conference and Exhibit, AIAA 2004-4724

[3] PANAGIOTOPOULOS,E.E., KYPARISSIS,S.D.: *CFD Transonic Store Separation Trajectory Prediction with Comparison to Wind Tunnel Investigations*, International Journal of Engineering, 3(6)

[4] LIJEWSKI,L.E., SUHS,N.E.: *Time-Accurate Computational Fluid Dynamics Approach to Transonic Store Separation Trajectory Prediction*, Journal of Aircraft, 31(4) (1994)

[5] HEIM,E.R.: *CFD Wing/Pylon/Finned Store Mutual Interference Wind Tunnel Experiment, Arnold Engineering Development Center*, AD-B152 669, September 10-17 1990

[6] Fluent Theory Manual, Version 2009, ANSYS, Inc., April 2009

[7] Fluent User's Guide, Version 2009, ANSYS, Inc., April 2009

[8] SUNAY,Y.E., GÜLAY,E., AKGÜL,A.: *Validation of CFD Simulation of Store Separation for EGLIN Test Case*, 5nd International Scientific Conference OTEH 2012, 18-19 september 2012., Belgrade, SERBIA

[9] GÜLAY,E., AKGÜL,A., ISAKOVIĆ,J., MANDIĆ,S.: *Computational Fluid Dynamics and Experimental Investigation of Wrap-Around-Fins Missile Rolling Moment*, Scientific Tehnical Review, ISSN 1820-0206, 2011, Vol. 61, No.3-4, pp.8-15.

[10] AGSARLIOGLU,E., AKGÜL,A.: *Numerical Prediction of Lateral Jets for Missile Like Geometries*, Scientific Tehnical Review, ISSN 1820-0206, 2012, Vol. 62, No.2, pp.3-9.

Received: 20.02.2013.

Numerička simulacija putanje odvajanja rakete od letelice primenom EGLIN testa

Jedan od najvažnijih problema u razvoju projektovanja raketa je bezbedno odvajanje rakete od letelice. Sami testovi odvajanja su skupi, vremenski zahtevni i opasni pošto sami testovi mogu biti sa fatalnim posledicama, stoga numeričke simulacije bezbednog odvajanja i predviđanja trajektorija rakete primenom numeričkih simulacija predstavlja kritični deo istraživanja. U ovoj studiji, korišćen je EGLIN test za verifikaciju predviđene trajektorije odvajanja od krila. EGLIN test je opšti model procesa odvajanja i gotovo redovno se koristi kao alat za verifikaciju trajektorija. Eksperimenti su bili razmatrani u transoničnom (M=0.95) i supersoničnim (M=1.2) domenima strujanja. EGLIN test model odvajanja je simuliran koristeći vremenski-zavisnu numeričku analizu dinamike fluida (CFD). Kombinacija

modela od šest stepeni slobode (6DOF) i solvera za strujanje je korišćena za predviđanje trajektorije odvajanja rakete od krila letelice. Za tu svrhu su korišćene, Navije-Stoksove i Ojlerove proračunske metode za uključivanje efekata viskoznosti fluida na trajektoriju kretanja rakete tokom odvajanja od krila.

Ključne reči: **numerička dinamika fluida, numerička simulacija, Navije-Stoksove jednačine, avionska raketa, putanja rakete, odvajanje rakete, EGLIN test.**

Численное моделирование траектории разделения ракеты от летательного аппарата применением EGLIN-теста

Одной из наиболее важных проблем в разработке ракет является безопасное отделение ракеты от летательного аппарата. Сами тесты разделения стоят дорого, являются трудоёмкими и опасными, ибо наши тесты могут быть с фатальными последствиями, поэтому численные моделирования безопасного разделения и прогнозирования траектории ракеты с использованием численного моделирования являются важной частью исследования. В этом исследовании мы использовали EGLIN-тест для проверки запланированной траектории отделения от крыла. При помощи EGLIN-тест возможно проверить общую модель процесса разделения и он почти всегда используется в качестве инструмента для проверки траектории. Эксперименты были рассмотрены в трансзвуковой (M = 0,95) и в сверхзвуковой (M = 1,2) областях потока. EGLIN-тест модели разделения был моделирован с помощью зависящих от времени численных анализов гидродинамики (CFD). Сочетание модели из шести степеней свободы (6DOF) и солвера (решателя) потока было использовано для прогнозирования траектории разделения ракеты от крыла летательного аппарата. Для этого использованы методы расчёта Навье-Стокса и Эйлера учитывая влияние эффекта вязкости жидкости на траекторию движения полёта ракеты во время отделения от крыла.

Ключевые слова: **численная гидродинамика, численное моделирование, уравнения Навье-Стокса, ракетоплан, траектория ракеты, реактивные разделения, EGLIN-тест.**

Simulation numérique de la trajectoire de séparation du missile de l'aéronef par le test EGLIN

La séparation en sécurité d'un missile de l'aéronef représente l'un des problèmes les plus importants dans le développement de la conception des missiles. Les tests même de la séparation sont coûteux, demandent beaucoup de temps et ils sont dangereux puisqu'ils peuvent avoir des conséquences fatales. C'est pourquoi les simulations numériques des séparations en sécurité ainsi que les prévisions des trajectoires des missiles par les simulations numériques représentent la phase critique des recherches. Dans cette étude on a utilisé le test EGLIN pour la vérification de la trajectoire prévue pour la séparation de l'aile. Le test EGLIN est un modèle général du processus de la séparation et il s'utilise régulièrement comme un outil pour la vérification des trajectoires. Les essais ont été considérés dans le domaine courants transsoniques (m = 0,95) et les supersoniques (m = 1,2). Le test modèle EGLIN a été simulé au moyen de l'analyse numérique de la dynamique des fluides (CFD) qui est dépendante temporellement. La combinaison du modèle de six degrés de liberté (6 DOF) avec le solveur pour les courants a été utilisée pour prévoir la trajectoire de séparation du missile de l'aile de l'aéronef . Dans ce but on a appliqué les méthodes de computation de Navier-Stokes et de Euler pour examiner les effets de la viscosité des fluides sur la trajectoire du missile lors de la séparation de l'aile d'aéronef.

Mots clés: **dynamique numérique des fluides, simulation numérique, équations Navier-Stokes, missile d'aéronef, trajectoire de missile, séparation de missile, test EGLIN.**