NUMERICAL SIMULATIONS OF WING DEPLOYMENT USING OVERSET GRID TECHNIQUE

Rakesh Kumar[1], Pranab Mondal[2] and K Selvaraj[2]

ABSTRACT

This study focuses on the numerical simulation of the unsteady three dimensional flow field during wing deployment of a typical air vehicle. Dynamic loading during wing deployment may play an important role in the stability and transition dynamics of the air vehicle. It may be noted that windtunnel or in-flight tests are usually difficult for this kind of study leaving no choice but to explore numerical techniques.

Wing deployment study has been carried out in past for the air vehicle under study using quasisteady approach and DATCOM based methods for aerodynamic characterization during deployment phase of flight. An attempt has been made to perform time-accurate simulation of wing deployment, using CFD methods utilizing Chimera technique. This paper highlights the Overset grid approach adopted to predict the unsteady force coefficients of the wing during deployment. A well-known test case [1], Generic-Wing-Pylon-Store (GWPS) configurations was chosen for validating the present study. Encouraging comparison was found with experimental data where trajectory was derived from force measurements during store transition. After validation, the methods were extended to a generic air vehicle configuration for prediction of flow during wing deployment. Deployment sequence.

Keywords: Wing Deployment, Unsteady Force Predictions, Store Separation, Overset Grid Technique

INTRODUCTION

Air vehicle undergoes transition in configuration during various phases of its flight. Aerodynamic characterization for these phases are essential to account for sudden changes of forces and moments, since configuration transition phases involves unsteady flow field around the air vehicle. Evaluating these unsteady aerodynamic characteristics could improve the accuracy of flight dynamics models. These models are important for the development of wing by fly-by-wire systems and autopilots, as well as for realistic flight simulation. This numerical simulations may be useful in designing wing deployment rate and mechanism. Generating such force and moment data either in wind tunnel or in-flight experiments are very difficult and expensive. Other option available to assess these unsteady effects is numerical simulation, which is the scope of this paper.

The present study analyses the unsteady flow features. The unsteady flow during the wing deployment of the wing body cruciform tail configuration of the air vehicle has been numerically simulated in order to assess differences to the steady state solution at extreme and intermediate wing positions which were carried out earlier through engineering methods. The primary objective of such analysis is to identify the effect of deployment time during wing opening. Extensive wind tunnel tests are needed to identify worst-case conditions that can be included in a minimalist set of flight test conditions. While wind tunnel tests could provide realistic data, it is practically infeasible to carry out such tests and would be increasingly expensive. With tremendous increase in computing power and the availability of cost effective clusters, CFD based predictions can drastically cut down this time and number of WT tests.

Unsteady aerodynamic data base using CFD during wing deployment can be generated either utilizing dynamic mesh or through overset mesh techniques. This paper describes the overset grid technique being used in conjunction with a 6DOF module for unsteady aero data predictions.

A well-known test case, Generic-Wing-Pylon-Store (GWPS) configurations was chosen for validating this study. The experiments were conducted at the Arnold Engineering Development Center at Tullahoma, TN [1] for several Mach numbers and angles of attack employing a Captive Trajectory System (CTS) technique to derive the store trajectory based on force measurements carried out during store movements. Since there are not any relevant wing/fin deployment validation case available in open literature, this was chose where trajectory was estimated based on force measurements during movements in conjunction with 6DOF, while performing experiments. In fact the wing motion also uses same 6DOF module whereby motions are restricted to represent rotations and hence same numerical techniques is deployed. Numerical results are compared with experimental results for M=0.95 and zero angle of attack. The second configuration is a generic canister launched cruise vehicle configuration. Preliminary studies were conducted for M=0.55 and at 2^0 angle of attack.

The primary objectives of this present study are

1) To gain confidence in Overset Grid Technique to accurately predict the unsteady forces and.
2) To apply this Overset Grid technique for Wing/Find employment application/ configuration ions.

SIMULATION APPROACH

The dynamic motion of the folded wing of a cruise vehicle during the deployment phase is quite complex. The current study focused on the numerical simulation of the unsteady three-dimensional flow filed around a generic cruise vehicle during the wing deployment. A commercially available compressible unsteady RANS solver with the Chimera technique has been utilized for this purpose. This methods is found to be suitable for the simulation of the flow around several bodies moving relatively to each other and already in use towards store separation studies. In general, to simplify the engineering analysis, the "Steady flow past a stationary object" assumption is often used and depending on the particular problem under study, this may be a reasonable approach. More often while approaching a store separation problem quasi-steady approach in-lieu of time-accurate computations are used. I.e. the component that is engaged in a relative motion is assumed to be instantaneously frozen.

To meet the primary requirements of wing deployment simulation and the flow around a wing during deployment sequence, the overset grid technique is utilized. Also the key to a successful wing unfolding and its motion prediction is to model the deployment process accurately with suitable kinematic and dynamic 3D models that simulate the geometry of wing motion in and out of the fuselage body. Extensive CFD is used to simulate the wing deployment phase of flight. The first and foremost requirement was to develop an extraction force model for simulation the pyro forces exerted on the wing towards unfolding process which allows varying forces with time representing the wing deployment process. The simulation of the unfolding wing requires a realistic model for applied loads acting on the folded wing. I.e. the time dependent force is needed as input for simulating this external forces during deployment apart from inherent computations of aerodynamic forces. Since model considered was only deployment angle vs. time, a representative simulation is carried out in the absence of force vs. time data.

The altitude at which this event takes place ranges between 500 and 700 m and at M=0.5-0.7. For current simulations the wing deployment altitude is taken as 600m at M=0.55 and at angle of attack of 2 deg. Computational simulations of wing deployment, involve a multi-step process from grid generation to setting up the model and obtaining a solution. The geometry becomes very complex with full cruise vehicle configuration with grooves and openings with stowed wing. Therefore, unstructured grids were utilized to build grid systems to define the complete domain in which calculations are to be performed. The following section briefly describes meshing technique.

OVERSET GRID TECHNIQUE

Overset grid technique also known as chimera technique is adopted for current simulations. In such technique, the computational domain is divided into two parts viz. Background Region & Overset Region. A typical schematic of overset domain is shown in Fig. 1. Overset meshes, overlapping meshes, are used to discretize a computational domain with several different meshes that overlap each other in an arbitrary manner. They are most useful in problems dealing with multiple or moving bodies, as well as optimization studies. In most cases, using overset meshes do not require any mesh modification after generating the initial mesh, thus offering greater flexibility over the standard meshing techniques. Any study involving overset meshes has a background region enclosing the entire solution domain and one or more smaller regions containing the bodies within the domain. In an overset mesh, cells are grouped into active, inactive, or acceptor cells. Within active cells, discretized governing equations are solved. Within inactive cells, no equation is solved, however, these cells can become active if the overset region is moving. Acceptor cells separate active and inactive cells in the background region and are attached to the overset boundary in the overset region. Acceptor cells are used to couple solutions on the two overlapping grids. Variable values at donor cells of one mesh express variable values at acceptor cells in the other mesh, through interpolation [2]. The donor cells are the active cells from the other mesh that are nearest the acceptor cell. In Fig. 1, one can see a typical overset grid topology and communication between two different grids. The solution is computed for all active cells in all regions simultaneously, that is, the meshes are implicitly coupled. When a reference is made to the variable value in an acceptor cell of one mesh within the discretized equations, a blend of variable values at donor cells from the other mesh is used. This value is reflected directly in the coefficient matrix of the algebraic equation system. This tight coupling of the overset and background regions allows for a solution that is within an arbitrary low level of iteration errors. The rate of convergence of the iterative solution method is therefore similar to that of a single mesh of the same resolution. Using this overset technique the test case configuration and generic air vehicle configuration are meshed and solved using RANS and URANS equations.

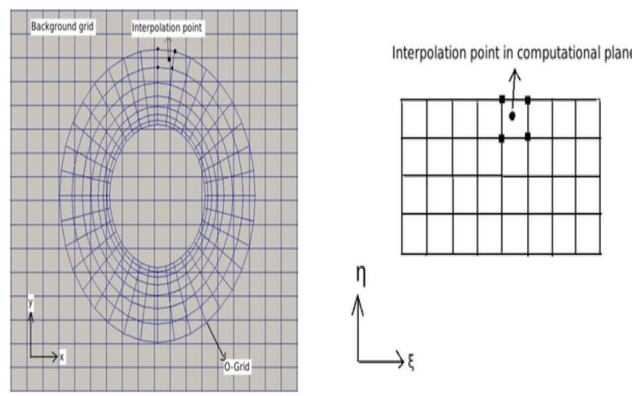
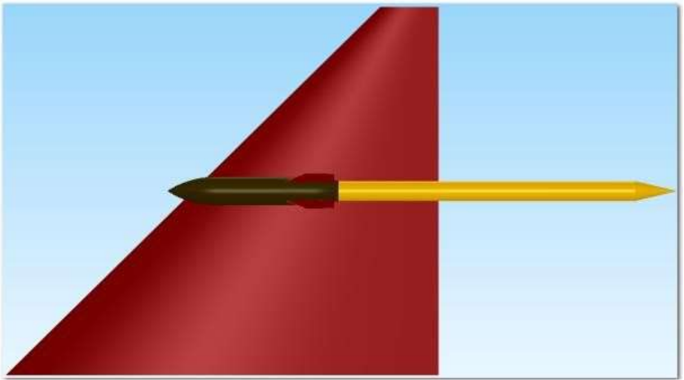


FIGURE 1: MESH TOPOLOGY FOR OVERSET TECHNIQUE [2]

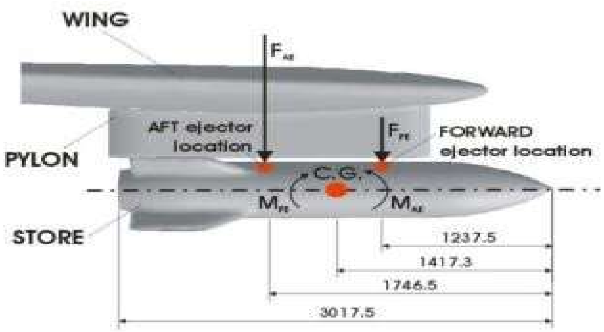
RESULTS & DISCUSSION

Test Case: EGLIN Benchmark (GWPS) Configuration

This configuration is used to validate the present approach of Overset grid technique. The test configuration [1] geometry consists of a clipped delta wing (NACA64A010 airfoil section) with a 45˚ leading edge sweep and a taper ratio of 0.133, a pylon located at the mid span of the wing, and a store having four tail fins positioned in a cruciform. The pylon has an ogive-flat plate cross section shape, which is closed at the leading and trailing edges by a symmetrical tangent- ogiveshape. The store is a cylindrical body with tangent give fore body. The store is modeled with the sting used in the CTS wind tunnel experiments to account for its effect on the flow field as this was not separated in the experimental processing of the data. The store has four fins that are located at the tail side in a cruciform. The fins are identical and they have a constant profile (NACA0008) throughout the fin span. Leading and trailing edge sweep angles of the fins are 60 and 0 degrees, respectively. The fins are clipped to the store body. The store is located at the mid-span of the wing at carriage position. The geometry of the configuration was built using the CAD properties of the pre-processor CFDGEOM. The surface generated through CFD-GEOM is shown in Fig. 2.



(a) CHEMATIC OF THE CONFIGURATION

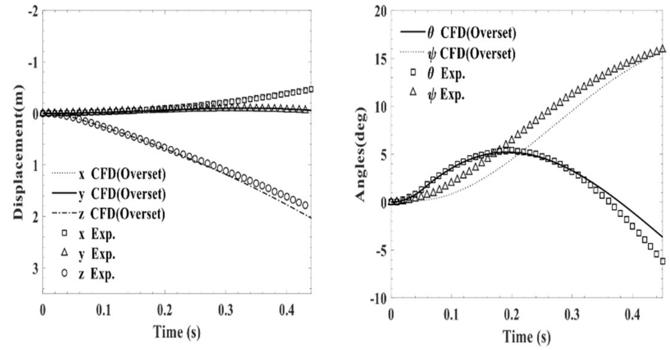


(b) EJECTOR FORCE MODEL USED IN THE EXP. [3]

FIGURE 2: EGLIN BENCHMARK CONFIGURATION (GWPS)

Time accurate Force/Trajectory Predictions & Comparisons with Experimental Data

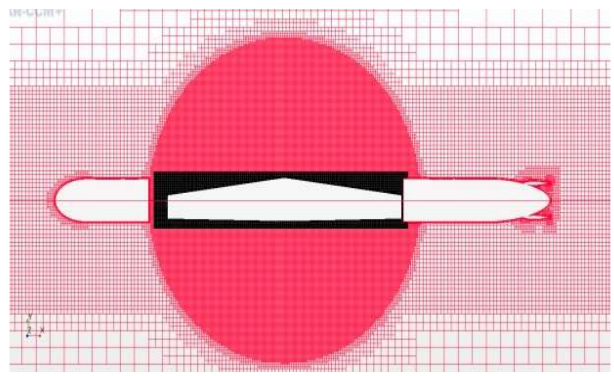
The trajectory is estimated based on quasi-steady force measurements and computations carried out for 0.95 Mach number at given flight condition and compared against experimental data available [1]. Linear and angular displacements graph are drawn with respect to the store's CG at the carriage location. From the Fig. 3, it is seen that the results show very good agreement with experimental data for linear displacements of the store from its initial location. Similar trends can be seen for linear and angular velocities and it has shown promising agreement with experimental values. Results shows that store pitches up, yaw to the outboard motion. Pitch up motion is mainly due to the ejector forces acting on the store for a real time of t=0.06sec. In the proximity region of the wing the motion is dominated by the external forces and moments applied by the ejector. And accurate model of the ejector were supplied through Ejector Modelling including offset moments. After vanishing of the ejector forces, the motion is dominated by aero dynamics force apart from inertial force. The effect of ejector forces can be clearly observed. Also, the flow regime is transonic for the benchmark configuration taken for CFD validation, the aerodynamic data (CFD) data need to be handled with more care. This problem may not occur at subsonic regimes for the current computations are carried out.



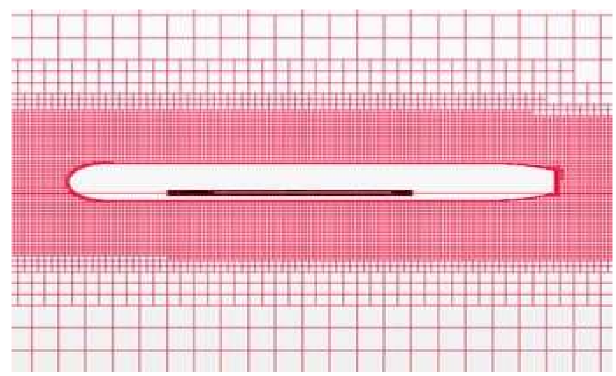
(a) Linear Displacements (b) Angular Displacements

FIGURE 3: COMPARISONS OF TRAJECTORY (M=0.95), PRESENT COMPUTATIONS (OVERSET GRID) VS EXPERIMENTAL DATA

GENERIC AIR VEHICLE CONFIGURATION



(a) SIDE VIEW OF THE VOLUME GRID



(b) TOP VIEW OF THE VOLUME GRID

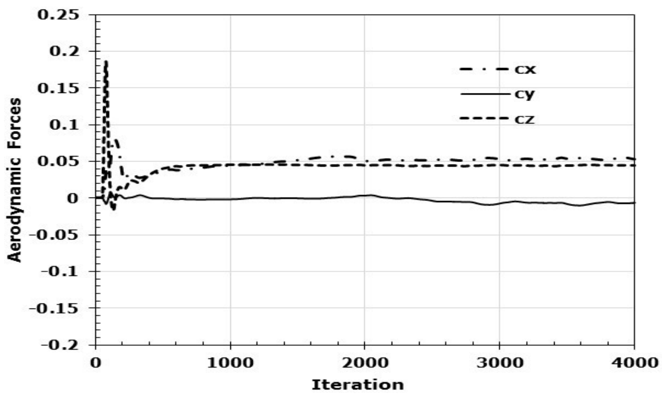
FIGURE. 4: COMPUTATIONAL GEOMETRY & GRID (AIR VEHICLE WING DEPLOYMENT)

Parameter	Generic Cruise Vehicle Configuration
Altitude (m)	600
Mach number	0.55
Angle of attack (deg)	2
Angle of Side Slip (deg)	0

TABLE 1: FREE STREAM FLOW QUANTITIES OF WING DEPLOYMENT SIMULATIONS

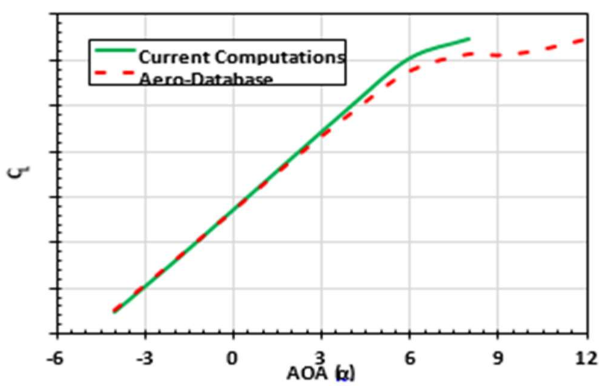
STEADY LIFT/DRAG PREDICTIONS FOR WING DEPLOYED CASE

Prior to computations of wing motion, the solution were obtained for the static forces & moments. The computations were performed for air vehicle configuration with wing fully deployed (without any groves & openings on fuselage to house the wing) at M=0.5 and compared against the existing aero-database (CFD based) used in the preliminary designs. The comparison of results (number removed intentionally) are shown in Fig. 5.



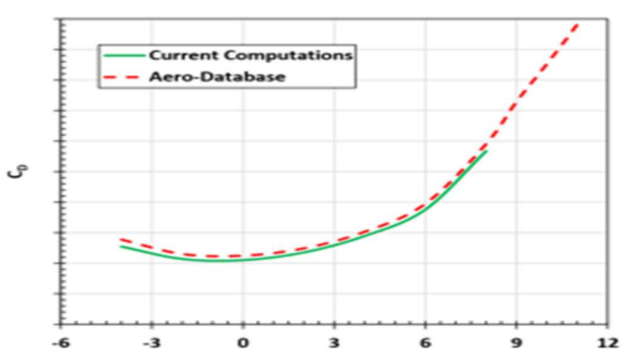
AOA (α)

(a) A TYPICAL FORCE CONVERGENCE HISTORY

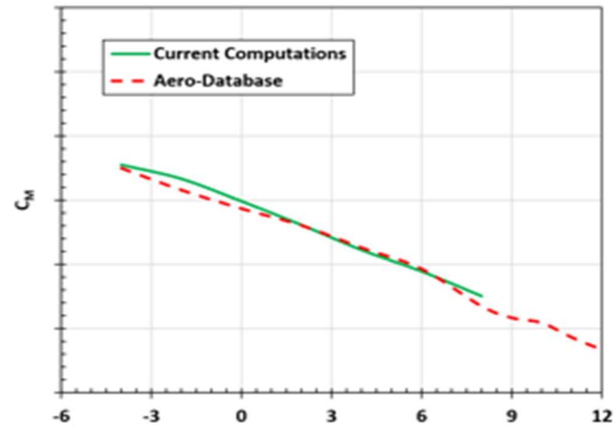


AOA (α

(b) COMPARISON OF LIFT COEFFICIENT



(c) COMPARISON OF DRAG COEFFICIENT

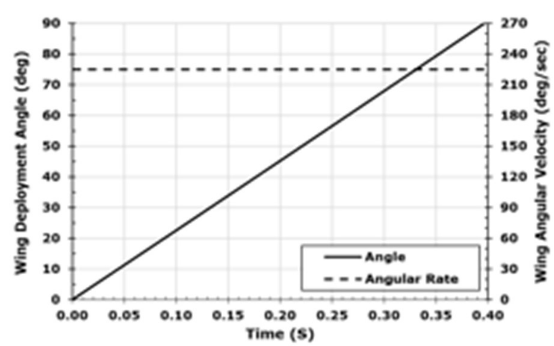


(d) COMPARISON OF MOMENT COEFFICIENT

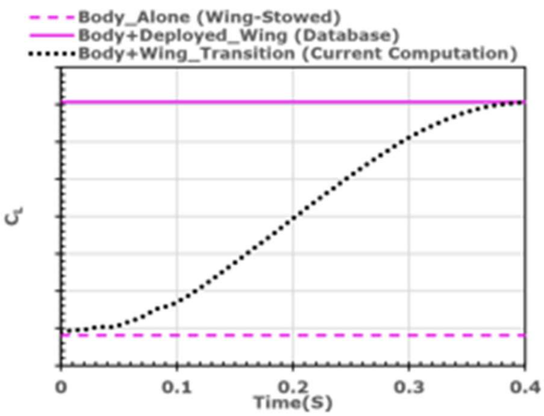
FIGURE 5: COMPARISON OF FORCE COEFFICIENT FOR AIR VEHICLE CONFIGURATION, M=0.5

TIME ACCURATE LIFT/DRAG PREDICTIONS FOR WING DEPLOYMENT CASE (TRANSITION PHASE)

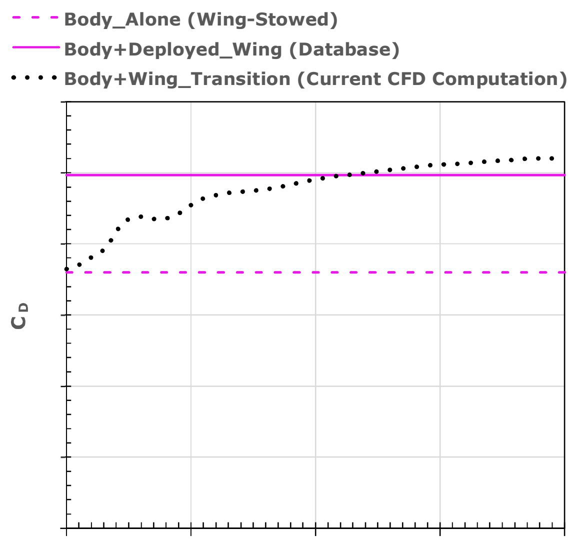
After comparisons of the results for wing fully deployed case, transient simulations were undertaken. Starting from the steady solutions (For M=0.55) on air vehicle Body Tail with wing in stowed position (with through openings in fuselage belly), transient solutions of the wing deployment were computed following the unfolding of the wing from the stowed position. Wing Pyro forces were simulated by giving a known yaw rate of the wing for simulation of deployment. A constant yaw rate was given as shown in Fig. 6a. The accuracy can be improved by supplying the external pyro forces as function of time simulating the deployment dynamics as demonstrated by Suk ET. Al [4]. Release Unit (RU) forces usually have a definite time profile and the same also can be modeled either as forces, initial velocities or prescribed motion applied to the wing. The RU forces has a significant impact on the initial motion of the wing which may not be linear.



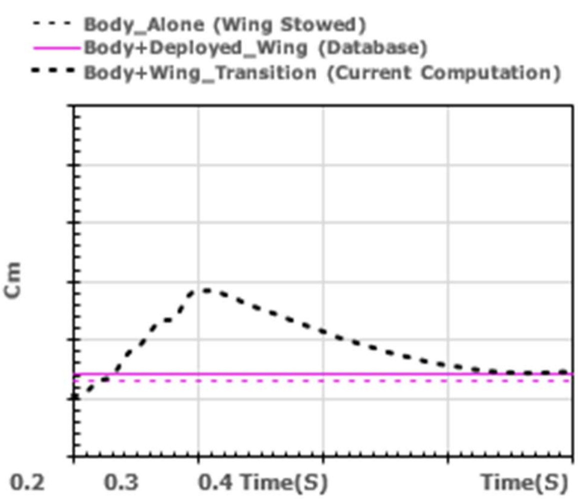
(a) SIMULATED WING DEPLOYMENT FORCE MODEL



(b) VARIATIONS OF LIFT COEFFICIENT/ LIFT BUILD-UP



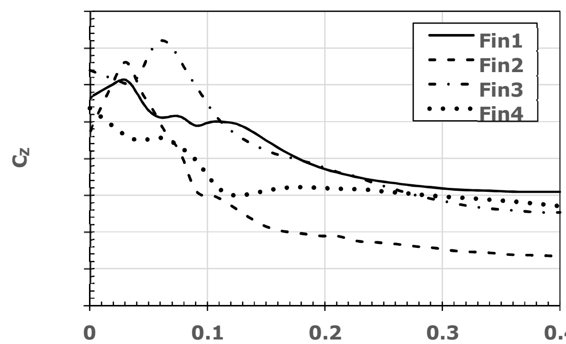
(c) VARIATION OF DRAG COEFFICIENT/ DRAG BUILD-UP



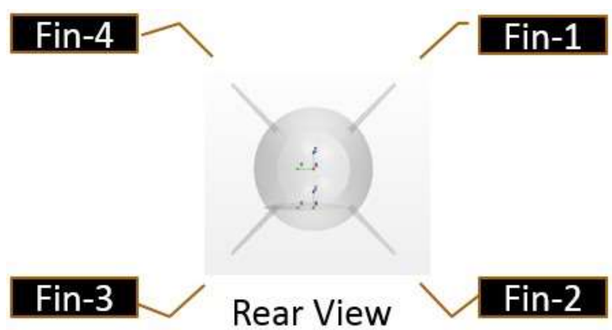
(d) Variation of Drag Coefficient/ Drag Build-up

FIGURE 6: COMPUTED FORCE BUILDUP, DURING WING DEPLOYMENT PHASE (M=0.55, α=2 deg, Alt=600m)

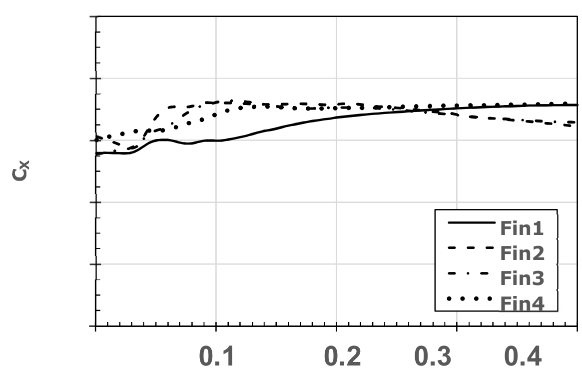
The computed forces and moments during ing deployment are shown in Fig. 6b - Fig. 6d. Simulations were performed for 0.4 seconds and the same is plotted. Parameters such as rolling and pitching moments during wing deployment are captured well. The interaction of Wing-Fin is also monitored during wing deployment by studying the forces on each fins. The fin-nomenclature followed is illustrated in Fig. 7a. Forces acting on each fin in all the three directions are shown in Fig. 7b -Fig. 7d. The flow visualization study were also carried out to understand the wing-fin interactions. Minor deviations can be attributed probably due to some physics not incorporated in the model such as the pyro-force model.



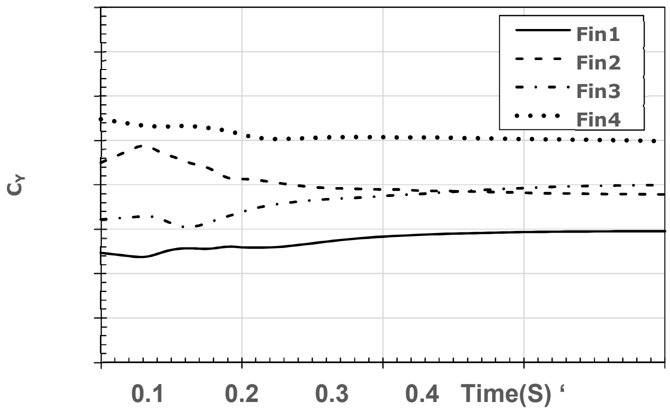
(a) NOMENCLATURE OF THE FINS (LOOKING FROM REAR)



(b) NORMAL-FIN FORCES



(c) AXIAL-FIN FORCES



(d) SIDE-FIN FORCES

FIGURE 7: COMPUTED UNSTEADY FORCES (M=0.6), DURING WING DEPLOYMENT PHASE

The computed results show a good match of lift coefficient at the end of deployment with steady state solution. The lift build up is captured during the wing transition from stowed to fully deployed condition. Scope exists to improve the current model by providing the proper input for modelling unfolding forces and moments. Accurate input of pyro forces would give a more realistic unfolding of wing under aerodynamic load as illustrated in Fig. 8. Clearly, unfolding yaw rate of the wing cannot be liner under severe aerodynamic loads. Since the data related to pyro model were not available, the methodology was developed with assumption for constant yaw rate for wing unfolding motion.

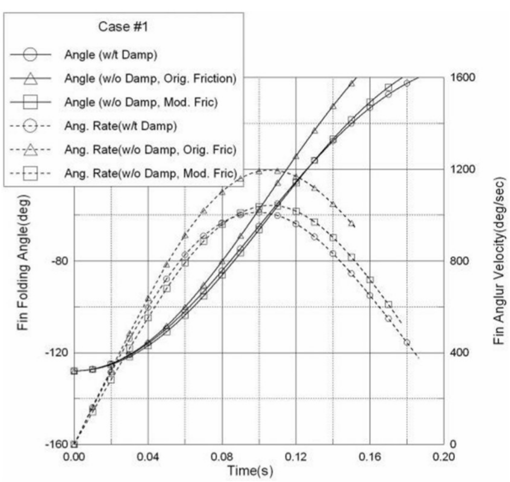


FIGURE 8: A LITERATURE DATA, WITH A REALISTIC AERODYNAMIC DAMPING MODEL [4]

CONCLUSIONS

The objective of the present work was to develop a modelling approach for the numerical simulation of wing deployment from a moving body. Wing deployment simulations are carried out and results are presented. The results are encouraging. The computational results predicted the magnitude of the aerodynamic wing un-folding moments to be considerably higher and more variable than initially estimated. The suggested aerodynamic modeling technique for the study of wing deployment is efficient and reliable enough to provide data for the design and analysis for the unfolding mechanisms for the air vehicle.

The same approach is also suitable for aerodynamic characterization for fin deployment and further study of wing and fin failure modes. Pyro force model needs to be incorporated to account for aerodynamic damping effects for numerical modelling of unfolding motion of the wing or fin.

REFERENCES

[1] Fox. J. H., 23. Generic Wing, Pylon, and Moving Finned Store, Verification and Validation Data for Computational Unsteady Aerodynamics, RTO-TR-26, October 2000, St. Joseph Ottawa/Hill, Canada.

[2] Rathakrishnan Bhaskaran and Sanjiva K. Lele. *"*Large eddy simulation of free-stream turbulence effect on heat transfer to a high-pressure turbine cascade." Journal of Turbulence, 11:N6, 2010.

[3] Elias E. Panagiotopoulos & Spyridon D. Kyparissis., CFD transonic store separation trajectory prediction with comparison to wind tunnel investigations, Mechanical Engineering & Aeronautics Department, University of Patras, Greece.

[4] Suk Young Jung & Sung Joon Yoon., "Aerodynamic Modeling of Folded Fin for Unfolding Motion Simulation", Journal of spacecraft and rockets, Vol. 47, No. 3, May-June 2010.