Computational Methods for Parachute Aerodynamics

T. Tezduyar[1]*, Y. Osawa[2], K. Stein[3], R. Benney[3], V. Kumar[1] and J. McCune[1]
[1]Team for Advanced Flow Simulation and Modeling (T*AFSM)
Mechanical Engineering and Materials Science
Rice University - MS 321, 6100 Main Street
Houston, TX 77005, USA
[2]Tire Research Department, Bridgestone Corporation
3-1-1 Ogawahigashi-cho, Kodaira-shi, Tokyo 187, JAPAN
[3]Natick Soldier Center, ATTN: AMSSB-RAD-T(N), Kansas Street
Natick, MA 01760-5017, USA

Summary

We highlight some recent methods developed by the Team for Advanced Flow Simulation and Modeling (http://www.mems.rice.edu/TAFSM/) for computation of parachute aerodynamics and fluid-structure interactions. This class of problems involve several computational challenges, including computation of unsteady long-wake flows generated by cargo aircraft carrying paratroopers and the affect of that unsteady wake on a parachute crossing it, as well as parachute aeromechanics simulations that take into account the changes in the parachute shape. Among the numerical methods we have developed to address these challenges are: a multi-domain method for computation of long-wake flows and flow around objects placed in such wakes, methods for the simultaneous solution of the fluid and structural mechanics equations governing the aeromechanics of a parachute, and advanced mesh moving methods. Our presentation here includes numerical examples that demonstrate the new computer simulation capabilities offered by the methods we have developed.

1. Introduction

In paratrooper deployment, typically, large number of paratroopers are deployed within a short time frame from multiple cargo aircraft flying in formation. This flight formation is based on the trailing aircraft flying, for obvious reasons, at a higher altitude relative to the leading one. The paratroopers deployed from the trailing aircraft sometimes descend through the unsteady wake of the leading aircraft. This wake is a substantial one, because while the paratroopers are being deployed, the aircraft slow down by bringing the wing flaps to their down positions. Although normally there is a large distance between the aircraft, the distance may not be large enough to spare the parachutes deployed from the trailing aircraft from being subjected to unsteady wake flows. Consequently, these wake flows need to be computed over long domains with sufficient accuracy to represent the actual vortex strength.

Computation of the aerodynamics of a parachute and determining its shape changes require solution of a coupled fluid-structure interaction problem. While the aerodynamic response of the parachute and the aerodynamic forces acting on it depend on the shape of the parachute, the deformation of the parachute depends on these aerodynamic forces. In this coupled problem, the fluid mechanics is governed by the Navier-Stokes equations of incompressible flows, and the structural mechanics is governed by the equations of motion applicable to a membrane. In our analysis, the membrane is assumed to undergo

*Corresponding author

M. Hafez et al., *Computational Fluid Dynamics for the 21st Century*

large displacements, and therefore although the membrane constitutive equations are based on linearly elastic behavior, the large displacements cause geometrically nonlinear behavior.

Changes in the shape of the parachute require that the fluid dynamics equations be solved with a method that can handle during the computations changes in the spatial domain. To this end, we use the Deforming-Spatial-Domain/Stabilized Space-Time (DSD/SST) finite element method [1,2] and its version developed for fluid-structure interactions [3]. In the DSD/SST method, the finite element formulation is written over the space-time domain of the problem, and this automatically takes into account the changes in (i.e. the deformations of) the spatial domain. The interpolation functions used are continuous in space and discontinuous in time. The discontinuity in time spares the method from leading to full-blown four-dimensional computations. As the spatial domain changes it shape, the fluid dynamics mesh is updated by using an automatic mesh moving method [4]. In this method, the motions of the nodes are governed by the equations of elasticity. It is essential that the main objective underlying the mesh moving method is to reduce the frequency of remeshing. Because, remeshing involves generation of new elements and nodes, and in complex 3D problems this typically requires automatic mesh generation, and this would lead to more time-consuming computations. Therefore, special emphasis is placed on development of mesh moving methods designed to reduce the deformation of the smaller elements. This is because the smaller elements are more prone to excessive deformation, and when these elements become too deformed, remeshing becomes necessary.

The Multi-Domain Method (MDM) [5] was developed for computation of unsteady long-wake flows behind a primary object and, in some cases, influence of this wake flow on a secondary object located in or crossing that wake. The MDM is based on dividing the complete simulation domain into an ordered sequence of overlapping subdomains. The inflow conditions for the first subdomain are extracted from the free-stream conditions. The flow field computed over a leading subdomain is used in specifying the inflow boundary conditions for the following subdomain. In parachute applications, the primary object is the leading aircraft, and the secondary object is a parachute deployed from the trailing aircraft. In this case the MDM is used to compute flow around the aircraft (in the first subdomain), flow around the parachute (in the last subdomain), and the long-wake flow (in the sudomains in between). The flow computations around the aircraft and in the long wake do not involve any changes in the spatial domain. Therefore these computations are based an a semi-discrete stabilized finite element formulation [6] with the streamline-upwind/Petrov-Galerkin (SUPG) [7] and pressure-stabilizing/Petrov-Galerkin (PSPG) [6] stabilizations. The computation in the last subdomain might involve parachute fluid-structure interactions and changing spatial domains. This would require the use of the DSD/SST method. If the computation in this last domain is for a parachute that is assumed to retain its shape, on the other hand, then the computations can be accomplished by simply using the semi-discrete formulation.

The methods described have been implemented for parallel computing using the MPI programming environment, and the results reported were obtained with parallel computations on the CRAY T3E-1200.

In Section 2, we review the governing equations. Brief reviews of the finite element formulations, fluid-structure coupling, the mesh update method, and the MDM are presented in Sections 3-7. The numerical examples are presented in Section 8, and the concluding remarks are provided in Section 9.