Applications in Airdrop Systems:
Fluid-Structure Interaction Modeling

Keith Stein*
Department of Physics, Bethel College
St. Paul, MN, 55112, USA
e-mail: k-stein@bethel.edu

Tayfun Tezduyar
Mechanical Engineering, Rice University – MS 321
6100 Main Street, Houston, Texas 77005, USA
e-mail: tezduyar@rice.edu

Richard Benney
Natick Soldier Center
Natick, MA 01760, USA
e-mail: Richard.Benney@natick.army.mil

Key words: flow simulation, parachute aerodynamics, fluid-structure interactions, moving meshes

Abstract
Parachute systems experience aerodynamic and fluid-structure interactions during all phases of operation. Substantial fluid-structure interactions can occur between two separate parachutes when they come close to each other. Fluid-structure interactions also play an important role in parachute operations during control-line pulls and releases, which are employed while maneuvering a parachute or during soft-landing. We present our methods for simulation of parachute aerodynamic and fluid-structure interaction behavior and demonstrate the capabilities of the method with results from a variety of recent airdrop application simulations.
1 Introduction

The performance of a parachute is strongly influenced by fluid-structure interactions and is also sometimes influenced by the aerodynamic and fluid-structure interactions of its canopy with other parachute canopies. In this paper, we describe our computational model for such interactions, and present recent results from a variety of simulations highlighting various types of parachute interactions. In the first case, our investigation focuses on fluid-structure interactions between the canopies of two separate parachutes coming close to each other. Previous studies have focused on purely aerodynamic interactions between two parachutes and on how the aerodynamic interactions depend on the horizontal distance between the parachutes [1]. Here, we study how such interactions are influenced when our computational model includes the fluid-structure interactions between the parachute canopy and the surrounding flow field. In the second case, we investigate the fluid-structure interactions of a single round parachute for a variety of control line inputs. Control-line pulls and releases are used for simulating induced gliding and soft-landing.

These simulations, in addition to providing preliminary results for different types of parachute aerodynamic and fluid-structure interactions, show how the computational methods described can be used for parachute applications in general. The interaction between the parachute canopy and the surrounding flow field is an essential component of a realistic parachute simulation, and thus the ability to predict parachute fluid-structure interactions is recognized as an important challenge within the parachute research community [2, 3, 4, 5, 6]. Follow-on studies will take more extensively into account the complex fluid-structure interactions involved at various stages of parachute systems, from initial deployment to landing and for single and clustered parachute systems.

For all simulations described in this paper, the parachutes are operating at sufficiently low speeds, and, therefore, the aerodynamics is governed by the Navier-Stokes equations of incompressible flows. In fluid-structure interactions, because the canopies undergo shape changes, the spatial domain occupied by the fluid is varying (i.e. deforming) with respect to time. Therefore we use the Deforming-Spatial-Domain/Stabilized Space-Time (DSD/SST) formulation [7, 8, 9], which was developed for flow problems with moving boundaries and interfaces.

2 Computational Model

2.1 Fluid Dynamics

Let \( \Omega_t \subset \mathbb{R}^{n \times d} \) be the spatial fluid mechanics domain with boundary \( \Gamma_t \) at time \( t \in (0, T) \), where the subscript \( t \) indicates the time-dependence of the spatial domain and its boundary. The Navier-Stokes equations of incompressible flows can be written on \( \Omega_t \) and \( \forall t \in (0, T) \) as

\[
\rho \left( \frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} - \mathbf{f} \right) - \nabla \cdot \mathbf{\sigma} = 0, \tag{1}
\]

\[
\nabla \cdot \mathbf{u} = 0, \tag{2}
\]

where \( \rho, \mathbf{u} \) and \( \mathbf{f} \) are the density, velocity and the external force, respectively. The stress tensor \( \mathbf{\sigma} \) is defined as

\[
\mathbf{\sigma}(p, \mathbf{u}) = -p \mathbf{I} + 2\mu \mathbf{\varepsilon}(\mathbf{u}). \tag{3}
\]
Here $p$, $I$ and $\mu$ are the pressure, identity tensor and the viscosity, respectively. The strain rate tensor is defined as
\[
\varepsilon(u) = \frac{1}{2}((\nabla u) + (\nabla u)^T).
\] (4)

Both Dirichlet- and Neumann-type boundary conditions are accounted for:
\[
u = g \text{ on } (\Gamma_t)_g, \quad n \cdot \sigma = h \text{ on } (\Gamma_t)_h.
\] (5)

Here $(\Gamma_t)_g$ and $(\Gamma_t)_h$ are complementary subsets of the boundary $\Gamma_t$, $n$ is the unit normal vector at the boundary, and $g$ and $h$ are given functions. A divergence-free velocity field is specified as the initial condition.

2.2 Structural Dynamics

Let $\Omega_s \subset \mathbb{R}^{n_{xd}}$ be the spatial domain bounded by $\Gamma_t$, where $n_{xd} = 2$ for membranes and $n_{xd} = 1$ for cables. The boundary $\Gamma_t^s$ is composed of $(\Gamma_t^s)_g$ and $(\Gamma_t^s)_h$. Here, the superscript “$s$” corresponds to the structure. The equations of motion for the structural system are:
\[
\rho^s \left( \frac{d^2y}{dt^2} + \eta \frac{dy}{dt} - f^s \right) - \nabla \cdot \sigma^s = 0,
\] (6)

where, $y$ is the displacement, $\rho^s$ is the material density, $f^s$ are the external body forces, $\sigma^s$ is the Cauchy stress tensor, and $\eta$ is the mass-proportional damping coefficient. The damping provides additional stability and is used for problems where time-accuracy is not important.

We use a total Lagrangian formulation of the problem. Thus, stresses are expressed in terms of the 2nd Piola–Kirchoff stress tensor $S$, which is related to the Cauchy stress tensor through a kinematic transformation. Under the assumption of large displacements and rotations, small strains, and no material damping, the membranes and cables are treated as Hookean materials with linear elastic properties. For membranes, under the assumption of plane stress, $S$ becomes
\[
S_{ij} = \left( \tilde{\lambda}_m G_{ij}^k G_{kl}^k + \mu_m \left[ G_{ij}^i G_{kl}^k + G_{ik}^i G_{jl}^j \right] \right) E_{kl},
\] (7)

where for the case of isotropic plane stress
\[
\tilde{\lambda}_m = \frac{2\lambda_m \mu_m}{(\lambda_m + 2\mu_m)}.
\] (8)

Here, $E_{kl}$ are the components of the Cauchy-Green strain tensor, $G_{ij}^k$ are the components of the contravariant metric tensor in the original configuration, and $\lambda_m$ and $\mu_m$ are Lamé constants. For cables, under the assumption of uniaxial tension, $S$ becomes
\[
S_{11} = E_c G_{11}^i G_{11}^i E_{11},
\] (9)

where $E_c$ is the cable Young’s modulus. To account for stiffness-proportional material damping, the Hookean stress–strain relationships defined by Eqs. (7) and (9) are modified, and $E_{kl}$ is replaced by $\dot{E}_{kl}$, where
\[
\dot{E}_{kl} = E_{kl} + \zeta \dot{E}_{kl}.
\] (10)

Here, $\zeta$ is the stiffness proportional damping coefficient and $\dot{E}_{kl}$ is the time derivative of $E_{kl}$.
2.3 DSD/SST Formulation of Fluid Dynamics

In the DSD/SST method, the finite element formulation of the governing equations is written over a sequence of $N$ space-time slabs $Q_n$, where $Q_n$ is the slice of the space-time domain between the time levels $t_n$ and $t_n+1$. At each time step, the integrations involved in the finite element formulation are performed over $Q_n$. The space-time finite element interpolation functions are continuous within a space-time slab, but discontinuous from one space-time slab to another. The notation $(\cdot)_n^-$ and $(\cdot)_n^+$ denotes the function values at $t_n$ as approached from below and above. Each $Q_n$ is decomposed into space-time elements $Q_n^e$, where $e = 1, 2, \ldots, (n,e)_n$. The subscript $n$ used with $n,e$ is to account for the general case in which the number of space-time elements may change from one space-time slab to another. The Dirichlet- and Neumann-type boundary conditions are enforced over $(P_n)_g$ and $(P_n)_h$, the complementary subsets of the lateral boundary of the space-time slab. The finite element trial function spaces $(S^h_0)_n$ for velocity and $(S^h_p)_n$ for pressure, and the test function spaces $(V^h_u)_n$ and $(V^h_p)_n = (S^h_v)_n$ are defined by using, over $Q_n$, first-order polynomials in both space and time. The DSD/SST formulation is written as follows: given $(u^h)_0^-$, find $u^h \in (S^h_u)_n$ and $p^h \in (S^h_p)_n$ such that $\forall w^h \in (V^h_u)_n$ and $q^h \in (V^h_p)_n$:

$$
\begin{align*}
\int_{Q_n} w^h \cdot \rho \left( \frac{\partial u^h}{\partial t} + u^h \cdot \nabla u^h - f^h \right) \, dQ + \int_{Q_n} \varepsilon(w^h) : \sigma(p^h, u^h) \, dQ \\
- \int_{(P_n)_h} w^h \cdot h^k \, dP + \int_{Q_n} q^h \nabla \cdot u^h \, dQ + \int_{\Omega_n} (w^h)^+ \cdot \rho \left( (u^h)^+_n - (u^h)^-_n \right) \, d\Omega \\
+ \sum_{e=1}^{(n,e)_n} \int_{Q_e} \frac{\tau_{LSME}}{\rho} \mathcal{L}(q^h, w^h) \cdot \left[ \mathcal{L}(p^h, u^h) - \rho \mathbf{f}^h \right] \, dQ \\
+ \sum_{e=1}^{(n,e)_n} \tau_{LSIC} \nabla \cdot w^h \rho \nabla \cdot u^h \, dQ = 0,
\end{align*}
$$

(11)

where

$$
\mathcal{L}(q^h, w^h) = \rho \left( \frac{\partial w^h}{\partial t} + u^h \cdot \nabla w^h \right) - \nabla \cdot \sigma(q^h, w^h),
$$

(12)

and $\tau_{LSME}$ (least-squares on momentum equation) and $\tau_{LSIC}$ (least-squares on incompressibility constraint) are the stabilization parameters (see [10]). This formulation is applied to all space-time slabs $Q_0, Q_1, Q_2, \ldots, Q_{N-1}$, starting with $(u^h)_0^- = u_0$. For an earlier, detailed reference on this stabilized formulation see [7].

2.4 Structural Dynamics Formulation

The semi-discrete finite element formulation for the structural dynamics is based on the principle of virtual work:

$$
\begin{align*}
\int_{\Omega_0^s} \rho^* \frac{d\mathbf{y}^h}{dt} \cdot \mathbf{w}^h \, d\Omega^s + \int_{\Omega_0^s} \eta \rho^* \frac{d\mathbf{y}^h}{dt} \cdot \mathbf{w}^h \, d\Omega^s \\
+ \int_{\Omega_0^s} \mathbf{S}^h : \delta \mathbf{E}(\mathbf{w}^h) \, d\Omega^s = \int_{\Omega_0^s} (t + \rho^* \mathbf{f}^s) \cdot \mathbf{w}^h \, d\Omega^s.
\end{align*}
$$

(13)
Here the weighting function $w^h$ is also the virtual displacement. The air pressure force on the canopy surface is represented by vector $t$. The pressure term is geometrically nonlinear and thus increases the overall nonlinearity of the formulation. The left-hand-side terms of Eq. (13) are referred to in the original configuration and the right-hand-side terms for the deformed configuration at time $t$.

Upon discretization using appropriate function spaces, a nonlinear system of equations is obtained at each time step. In solving that nonlinear system with an iterative method, we use the following incremental form:

$$
\left[ \frac{M}{\beta \Delta t^2} + \frac{(1 - \alpha) \gamma C}{\beta \Delta t} + (1 - \alpha) K \right] \Delta d^i = R^i,
$$

where

$$
C = \eta M + \zeta K.
$$

Here $M$ is the mass matrix, $K$ is the consistent tangent matrix associated with the internal elastic forces, $C$ is a damping matrix, $R_i^i$ is the residual vector at the $i^{th}$ iteration, and $\Delta d^i$ is the $i^{th}$ increment in the nodal displacements vector $d$. In Eq. (14), all of the terms known from the previous iteration are lumped into the residual vector $R^i$. The parameters $\alpha, \beta, \gamma$ are part of the Hilber–Hughes–Taylor [11] scheme, which is used here for time-integration.

### 2.5 Mesh Update Method

How the mesh should be updated depends on several factors, such as the complexity of the moving boundary or interface and overall geometry, how unsteady the moving boundary or interface is, and how the starting mesh was generated. In general, the mesh update could have two components: moving the mesh for as long as it is possible, and full or partial remeshing (i.e., generating a new set of elements, and sometimes also a new set of nodes) when the element distortion becomes too high.

In mesh moving strategies, the only rule the mesh motion needs to follow is that at the moving boundary or interface the normal velocity of the mesh has to match the normal velocity of the fluid. Beyond that, the mesh can be moved in any way desired, with the main objective being to reduce the frequency of remeshing. In 3D simulations, if the remeshing requires calling an automatic mesh generator, the cost of automatic mesh generation becomes a major reason for trying to reduce the frequency of remeshing. Furthermore, when we remesh, we need to project the solution from the old mesh to the new one. This introduces projection errors. Also, in 3D, the computing time consumed by this projection step is not a trivial one. All these factors constitute a strong motivation for designing mesh update strategies which minimize the frequency of remeshing.

In some cases where the changes in the shape of the computational domain allow it, a special-purpose mesh moving method can be used in conjunction with a special-purpose mesh generator. In such cases, simulations can be carried out without calling an automatic mesh generator and without solving any additional equations to determine the motion of the mesh. One of the earliest examples of that, 2D computation of sloshing in a laterally vibrating container, can be found in [7]. Extension of that concept to 3D parallel computation of sloshing in a vertically vibrating container can be found in [12].

In general, however, we use an automatic mesh moving scheme [13] to move the nodal points, as governed by the equations of linear elasticity, and where the smaller elements enjoy more protection from mesh deformation. The motion of the internal nodes is determined by solving these additional equations, with the boundary conditions for these mesh motion equations specified in such a way that they match the
normal velocity of the fluid at the interface. In computation of fluid-structure interactions of parachute systems reported here we use this automatic mesh moving technique.

3 Numerical Examples

For fluid dynamics equations we use tetrahedral meshes. The fluid-structure interaction examples we present utilize a model representative of a T–10 personnel parachute. Examples from earlier work, for cases where only aerodynamic interactions were considered, are based on a C–9 parachute model. The simulations are carried out at a Reynolds number (based on the canopy diameter) of approximately 5 million.

3.1 Fluid-Structure Interactions of Two Parachutes

Parachutes can encounter a variety adverse flow fields which produce strong fluid-structure interactions. The adverse flow fields could be due to winds (gusts, shear), aircraft wakes, and flow fields from nearby parachutes. Parachutes can also encounter adverse flows from from nearby parachutes or from canopies within parachute clusters [1]. We focus on the fluid-structure interactions of a parachute due to the adverse flow field of a nearby parachute. In this simulation, the two parachutes have an initial horizontal spacing of 42 ft (approximately 3 inflated radii) and a vertical spacing of 56 ft. Here, the parachute model is representative of a standard US Army T–10 personnel parachute. The T–10 is a “flat extended skirt canopy” composed of a 35–foot diameter canopy and 30 suspension lines each 29.4 ft long. The canopy is called a “flat extended skirt canopy” because in its constructed (or unstressed) configuration it is composed of a main circular section with a circular vent at the apex and an inverted flat ring section, which lies under the main section and is connected to the main section at the outer radius. The lines connect to four risers which attach the payload (or paratrooper). The suspension lines continue as 30 gore–to–gore reinforcements through the parachute canopy and meet at the apex. For the T–10, the vent diameter and the skirt width are both 3.5 ft.

Here the lower canopy is treated as a rigid body, while the upper canopy is allowed to deform due to the response of the parachute structure to the fluid dynamics forces. The structural dynamics model is divided into six distinct material groups; a membrane group, three cable groups, a truss group, and a concentrated mass group. The parachute canopy is composed of 780 biquadratic membrane elements. We have distinct cable groups for the suspension lines, the canopy radial reinforcements, and the risers. The truss and concentrated mass groups define the payload, which has a total weight of 250 pounds. The structure is allowed to fall completely unconstrained during the fluid-structure interaction computation.

The parachute canopies are represented as interior surfaces in the fluid mesh (with 17,490 triangular faces on both the upper and lower canopy surfaces). The typical size of the volume mesh is approximately 3.5 million elements and 580 thousand nodes, resulting in approximately 4.6 million coupled equations with the DSD/SST formulation. The automatic mesh update method described earlier is employed to handle the canopy shape changes, with occasional remeshing of the fluid domain. The surface for the upper canopy is assigned a no-slip boundary condition, with velocities coming from the structural dynamics solution. The boundary conditions for the lower canopy and at the outer boundaries are identical to the conditions used in the previous example. The coupling is achieved iteratively, by transferring the information between the fluid and structure with a least-squares projection. The vorticity field surrounding the two parachutes and the dynamics of the parachute structure at several instants during the simulation are
shown in Figures 1 and 2. Figure 3 shows, for the purpose of comparison, the aerodynamic interactions between two C-9 parachutes, predicted from a purely aerodynamic simulation reported earlier [1]. The first frame in Figure 1, which corresponds to the start of the fluid-structure interactions, shows qualitative similarities to the purely aerodynamic interactions depicted in Figure 3. The second and third frames, however, demonstrate the strong structural response of the upper parachute and onset of canopy collapse due to the interaction with the adverse flow field.

![Figure 1: Fluid-structure interactions between two parachutes (time=0-3.5 seconds). Vorticity.](image)

![Figure 2: Fluid-structure interactions between two parachutes (time=0-3.5 seconds). Structural deformation, with dark regions of the canopy depicting high differential air pressure.](image)

3.2 Fluid-Structure Interactions During Control-Line Pulls and Releases

The response of a parachute system to control-line inputs arises in many airdrop applications. One example involves the extension of single or multiple risers to achieve guidance for steerable round parachute
systems such as the Advanced Guided Airdrop System (AGAS) [14]. Here, control-line releases provide a round parachute system with minimal glide performance. A second example involves response of personnel parachutes to one- and two-riser “slips,” where control lines (risers) are pulled to provide final adjustments to paratrooper orientation just prior to landing. A third example involving control-line inputs is a parachute with a soft-landing retraction system. In these applications, a control-line or muscle attached between the payload and suspension lines provides a rapid contraction at landing, resulting in reduced landing impact. In each of these situations, the response of the parachute is strongly influenced by fluid-structure interactions. We present results from fluid-structure interactions simulations for each of these three types of control-line operations.

Preliminary simulations have been carried out to predict the response of a T–10 parachute system to a released riser (Figure 4) and to a dynamic single-riser slip (Figure 5). In each case, the dark regions on the canopy represent areas of highest differential air pressures. For the simulation involving the released riser, the structural model for the T–10 parachute is modified so that three of the risers have an unstressed length of 2.5 ft and the fourth riser has an unstressed length of 5.5 ft, representing a 3-ft riser release. The frames show the structural response of the parachute as it interacts with the surrounding flow field. At the end of the computation, the onset of gliding behavior is apparent as the parachute begins to pitch due to the release of the riser. The computation is terminated at the onset of membrane-to-membrane contact in the parachute canopy. Sophisticated nonlinear and dynamic contact algorithms are being developed that can handle this type of membrane-to-membrane contact. For the simulation involving the dynamic riser slip, each of the four risers has initial unstressed lengths of 2.5 ft. A riser slip is represented by reducing the unstressed length for one of the risers by approximately 2 ft in 0.7 seconds during the fluid-structure interaction simulation. The frames in Figure 5 show the structural behavior of the parachute as it responds to the riser slip and interacts with the surrounding flow field. Again, the computation is terminated at the onset of membrane-to-membrane contact.

In the next set of simulations, the soft landing of a T–10 parachute is modeled with different control-line inputs. Here, the structural model for the T–10 parachute is modified so that the suspension lines meet at a single confluence point and the payload is represented by a single lumped mass. A 14-ft pneumatic muscle actuator (PMA) is modeled with 10 cable elements that connect the confluence point and the payload. Figure 6 shows the behavior of a soft-landing system for a T–10 parachute and payload. The
sequence of photographs were obtained from a test conducted by the US Army at Fort Benning for a soft-landing system designed by Vertigo, Inc. This system employs a PMA attached between the payload and suspension lines, which provides a rapid contraction at landing, resulting in a reduced landing impact [14].

Three simulations are carried out for the soft-landing of a T–10 parachute system. In each case, the total parachute system weight is 300 pounds and is initially falling at the rate of approximately 20 ft/s. Contraction of the PMA is modeled by shrinking the cables during the soft-landing simulation. The shrinking of cables, or retraction, reduces the natural length of the PMA by 38 percent of its initial length during the simulation. Fluid-structure interaction computations are carried out for three different retraction times. In each case, a computation of 400 time steps is carried out prior to initiating the soft-landing. Then, shrinking of the PMA cables are prescribed to simulate contraction times of 0.14 seconds (200 time steps), 0.21 seconds (300 time steps), and 0.28 seconds (400 time steps). The structural model for the T–10 parachute system with the PMA, the fluid mesh, and the initial flow field are shown in Figure 7. Figure 8 shows, for each case, the aerodynamic drag experienced by the parachute canopy prior to, during, and immediately after the PMA contraction. As expected, the curve with the largest drag corresponds to the fastest retraction. Figure 9 shows, for each case, the vertical positions and velocities of the payload and confluence point during the same time interval. The soft-landing behavior is evident from the vertical velocity histories of the payload. For each of the cases simulated, the retraction results in a negative velocity for the payload shortly after the retraction has completed.
4 Concluding Remarks

We have described our computational methods for simulation of a variety of parachute aerodynamic and fluid-structure interactions. We considered two different types of problems. In the first case, we focused on fluid-structure interactions between the canopies of two separate parachutes in close proximity to one another. Earlier studies focused on aerodynamic interactions between two parachutes for different horizontal spacings and showed significant interactions when the horizontal spacing between the parachutes is two canopy radii or less. Here, we studied how the interactions between the two parachutes are influenced when we include in our computational model the fluid-structure interactions. The significant amount of structural response we observe in this study for the upper parachute makes it clear that fluid-structure interactions play a key role in making this class of simulations more realistic. In the second
Figure 8: Soft-landing of a T–10 parachute with PMA. Aerodynamic drag.

Figure 9: Soft-landing of a T–10 parachute with PMA. Position and velocity of the payload and confluence point.

case, we focused on the fluid-structure interactions for a single round parachute with a variety of control line inputs. The control line inputs investigated included pulls and releases to simulate induced gliding and soft-landing.

It is evident that fluid-structure interactions play a major role in the class of simulations presented in this paper. In these cases, sophisticated fluid-structure interaction models are required to accurately represent the response of the parachute structure.
Acknowledgements

This work was supported by the Army Natick Soldier Center and NASA JSC.

References


