Computing in Science & Engineering, IEEE, **January/February** (2003) 39-46.

COMPUTATIONAL METHODS FOR MODELING PARACHUTE SYSTEMS

Using computational models in parachute system development can improve performance. For successful modeling, however, several challenges must be addressed, particularly the interaction between the parachute structural dynamics and the aerodynamics.

The development of computational models that predict the aerodynamic performance of parachute systems has significantly enhanced the technology of airdrop systems by providing alternatives to or complementing drop tests and laboratory experiments. Increases in the scope and reliability of computational modeling are bringing current technology closer to establishing "virtual proving ground" models for a wide range of parachute applications such as personnel and cargo parachutes.

Traditional methods for developing airdrop systems rely heavily on full-scale testing, which can be prohibitively expensive, time-consuming, and limited in the data they provide. Computational modeling can complement these methods. The primary challenge in modeling parachute systems is representing the coupling between the airflow around the parachute and the parachute's structural dynamics. In almost all cases, this coupling plays a major role in parachute performance and in accurately representing the parachute's complex dynamics in a model requires that we treat the problem as a fluid–structure interaction (FSI) problem. We cannot obtain solutions for the parachute flow field and structural dynamics independently; rather, we need an algorithm that lets us match shared fluid and structural information (that is, the parachute displacements and velocities on one side and the aerodynamic forces on the other). For this reason, we need suitable techniques for fluid and structural computations. We use the Deforming-Spatial-Domain/Stabilized Space–Time (DSD/SST) formulation[1] to develop FSI models, and base our structural dynamics computations on a Lagrangian finite-element formulation for a cable–membrane tension structure.[2]

In this article, we highlight methods recently developed by Rice Univerity's Team for Advanced Flow Simulation and Modeling (www.mems.rice.edu/TAFSM) for parachute computations, such as DSD/SST and advanced mesh update methods. We focus on the challenges involved in developing computational models of airdrop systems and the computational issues that arise when simulating interactions between parachute structural dynamics and aerodynamics.

1521-9615/03/$19.00 © 2003 IEEE
Copublished by the IEEE CS and the AIP

KEITH STEIN
Bethel College
TAYFUN TEZDUYAR
Rice University
RICHARD BENNEY
Natick Soldier Center

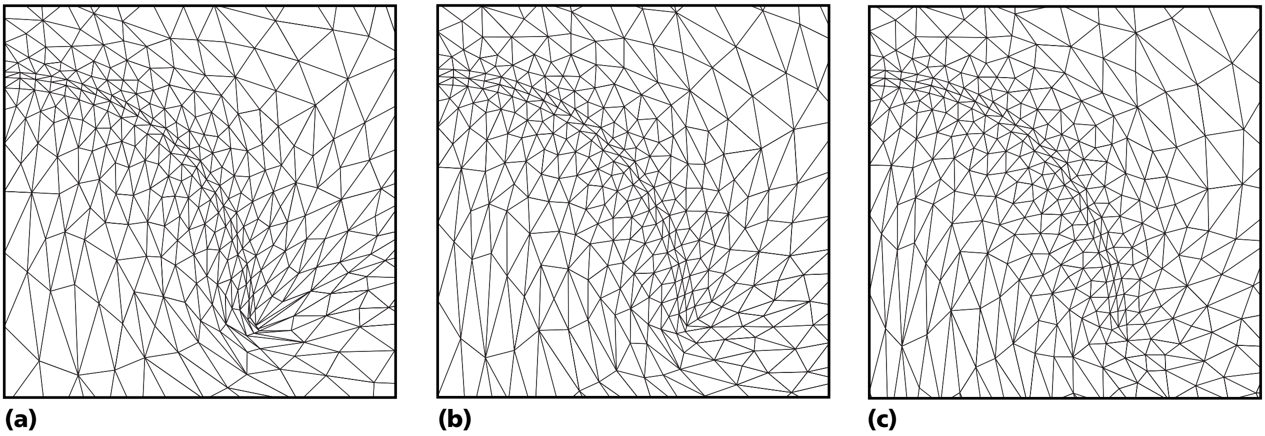

Figure 1. Jacobian-based stiffening (JBS) bending tests using a deformed mesh for $\chi = 0.0$, in which (a) no stiffening occurs; $\chi = 1.0$, in which (b) the stiffening is proportional to the inverse of the Jacobian; and $\chi = 2.0$, in which (c) stiffening is proportional to the inverse of the Jacobian squared.

Parachute aerodynamics and structural dynamics

Because the features of the complex and unsteady flows surrounding a parachute canopy depend on the canopy's shape, which in turn depends on the flow field, simulating the time-dependent parachute FSI is a difficult task. Parachute behavior is one of the "grand challenges" in FSI modeling because the aerodynamic structure changes shape significantly from one operation stage to the next. For example, during inflation, a parachute changes from a tightly packed bag into an inflated canopy in seconds. Only coupled FSI models can attempt to represent this behavior. Modeling other stages of parachute operation—terminal descent, maneuvering, interaction between multiple parachutes, and so on—faces the same challenge.

Challenges in FSI modeling

To successfully model parachute systems, we must first address several computational challenges, such as flows with changing spatial (mathematical) domains, mesh management, and strong coupling between the airflow around a parachute and the parachute structural dynamics.

Flows with Changing Spatial Domains

To compute flow problems with moving boundaries and interfaces, we can use *interface-tracking* or *interface-capturing* methods.[3] The method we use depends on the problem being addressed. Interface-capturing methods represent the interface in a fixed domain through an interface function. The interface is captured within the resolution of the finite-element mesh covering the area in which the interface is located. Interface-tracking methods follow the interface and precisely represent it as a surface contained in the finite-element mesh. This method is more suitable for accurate computation of the boundary layers near solid surfaces (that is, the parachute canopies). In an interfacing-tracking method, the mesh needs to be updated during the computations as the solution evolves.

The DSD/SST formulation suits our modeling challenges. This formulation, which was developed for flow problems with moving boundaries and interfaces, is an interface-tracking method capable of handling parachute shape changes within the fluid domain. In the DSD/SST method, because the finite-element formulation of the problem is written over the associated space–time domain, the motion of moving interfaces is automatically taken into account. This method is based on stabilization techniques, namely the Streamline-Upwind/Petrov-Galerkin (SUPG)[4] and Pressure-Stabilizing/Petrov-Galerkin (PSPG)[1] formulations. Stabilization techniques let us compute, without numerical instabilities, flows in realistic flow speed ranges. We can also use equal-order interpolation functions for velocity and pressure. This has several advantages, including easier parallel implementation.

Mesh Management

In parachute FSI computations, the parachute canopy surface is an embedded interface in the fluid mesh that tracks the canopy motion, and we therefore need suitable mesh management techniques. For simple problems and interface geometries, we can use special-purpose mesh-generation and mesh-moving techniques. Such techniques can be efficient, but their use is limited.

Because most real-world problems, such as parachute FSI, involve complex geometries and arbitrary motions, we use mesh update methods consisting of mesh-moving and remeshing as needed. Remeshing involves additional costs for generating mesh and projecting the solution from the old mesh to the new one. It also introduces projection errors. Mesh motion requires the mesh's normal velocity to match the fluid's normal velocity at the interface. For problems involving large displacements, remeshing is unavoidable, so the objective would be to develop methods that minimize remeshing frequency. To that end, we use mesh-moving techniques[5] in which the elasticity equations govern internal node motion.

We treat the mesh selectively based on element sizes by modifying the Jacobian of the transformation from the element domain to the physical domain.[5] This Jacobian-based stiffening (JBS) method stiffens the smaller elements proportionally to the Jacobian inverse. We generalize JBS by introducing a stiffening power (χ) that determines how much stiffer to render the smaller elements than the larger ones.[6] Figure 1 shows the deformed mesh for stiffening powers ranging from 0.0 to 2.0 for a test case in which the interface deforms from a straight line to a half-circle over 50 increments. In this example, the increased stiffening clearly reduces mesh distortion significantly, especially near the interface tips.

The Solid-Extension Mesh-Moving Technique (SEMMT)[7,8] was developed for moving the mesh in fluid–solid interface computations where the fluid mesh has layers of thin elements adjacent to the interface. SEMMT treats those layers of elements like an extension of the solid elements. It does this in two ways:

- SEMMT—Multiple Domain solves elasticity equations for the nodes connected to the thin elements separately from the elasticity equations for the other nodes.
- SEMMT—Single Domain solves the elasticity equations for the nodes connected to the thin elements together with the equations for the other nodes.

When SEMMT solves the thin elements separately, it specifies a traction-free boundary condition at the interface with the other elements. When SEMMT solves them together with other fluid elements, it assigns the thin elements a higher rigidity than the other elements. We reported a several two-dimensional test cases for the SEMMT elsewhere.[9]

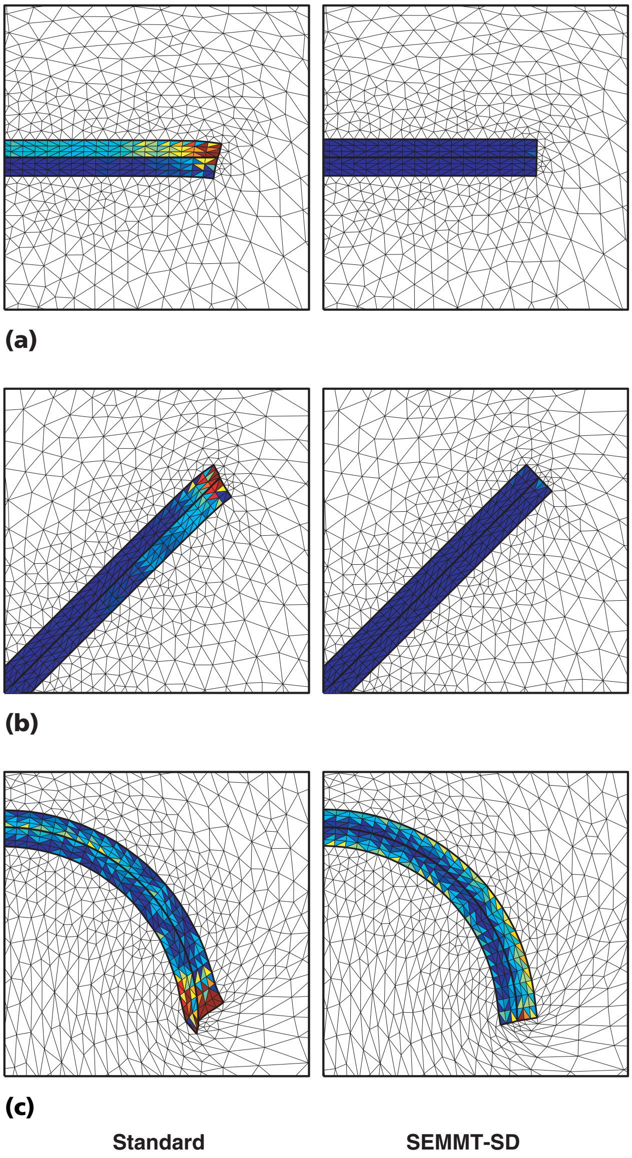

Figure 2. Mesh deformation tests with the standard mesh-moving technique and SEMMT—Single Domain. Three types of prescribed interface motion has been tested: (a) rigid-body translation, (b) rigid-body rotation, (c) and pure bending.

Figure 2 compares deformed meshes for the standard mesh-moving technique[5] and SEMMT—Single Domain. The figure shows close-up views of the deformed mesh for three types of prescribed interface motion: rigid-body translation, rigid-body rotation, and pure bending. As in Fig-

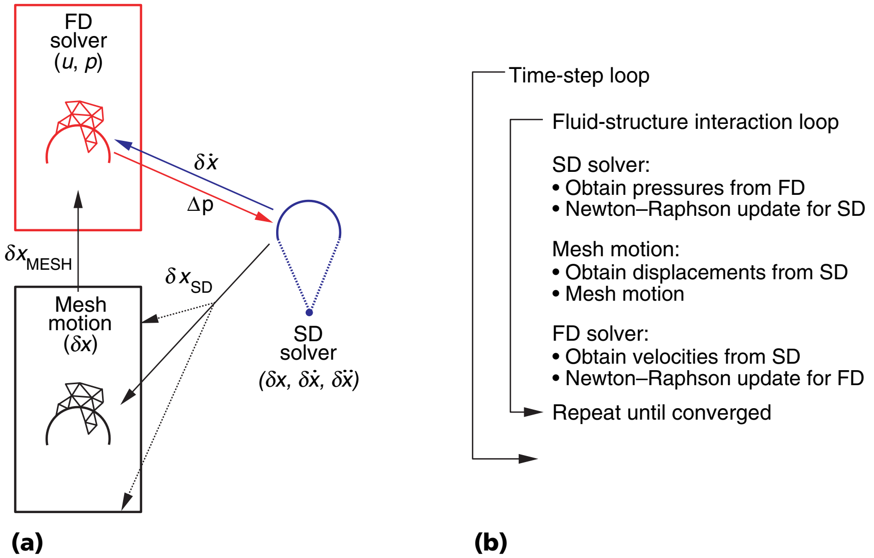

Figure 3. Iterative coupling strategy. We successively update (a) solution vectors for fluid dynamics (FD), structural dynamics (SD), and mesh motion (MM). Updates take place within a (b) nonlinear iteration loop embedded in a time-step loop.

ure 1, the mesh motions are prescribed over 50 increments. The inner element colors depict element-area distortion, with blue and red corresponding to low and high distortion, respectively. Clearly, SEMMT handles the prescribed interface motions with significantly less distortion of the thin elements.

Fluid–Structure Coupling

At every time step, we use an approximate Newton–Raphson method to solve the large, coupled nonlinear equation systems for fluid dynamics (FD), structural dynamics (SD), and mesh motion (MM). This method is based on iterative coupling. We successively update the solution vectors corresponding to each block (FD, SD, and MM) at each nonlinear iteration step. Data transfer between blocks takes place prior to the individual updates. Figure 3 illustrates this iterative coupling method. Figure 3a shows the successive updates for the FD, SD, and MM, with the arrows indicating the direction of information transfer. For example, the SD solver passes parachute canopy displacements (δx_{SD}) and displacement rates ($\delta\dot{x}$) for use as boundary conditions in the MM and FD solvers, respectively. The FD solver passes differential pressures (Δp) on the canopy to the SD. We use the updated mesh to solve the flow equations. Figure 3b shows the nonlinear iteration loop in which successive updates occur. The FSI loop is embedded in a time-step loop.

Simulations and Test Computations

We focus on three classes of simulations: multiple parachute interactions, parachute control-line inputs, and steerable round parachutes. The size of our equation systems and the lengthy computations necessitates the use of parallel supercomputers. We performed the simulations described in this article on a 1,056-processor Cray T3E-1200 and a 41-node Pentium IV PC cluster.

Multiple Parachute Interactions

Strong fluid–structure interactions can occur when parachute systems encounter adverse flow fields, such as wind gusts or wind shear, aircraft wakes, and flow fields of nearby parachutes. They can occur

- Between two parachutes, when one enters the wake of the other
- Between the canopies of a cluster of parachutes

We have performed computations to study both interaction types.

Between two parachutes. Two parachutes can interact when multiple paratroopers or payloads are deployed in a short timeframe. Initial simulations focused on the purely aerodynamic interactions between two parachutes, treating the parachute canopies as rigid, nondeforming geometries.[10] In these simulations, we studied the influence of parachute separation distance on the aerodynamic interactions. More recent studies have focused on the FSI behavior between two separate parachutes.[11,12]

Here we investigate the FSI of two parachutes with given initial relative positions.[10] The two parachutes start with a horizontal spacing of 42

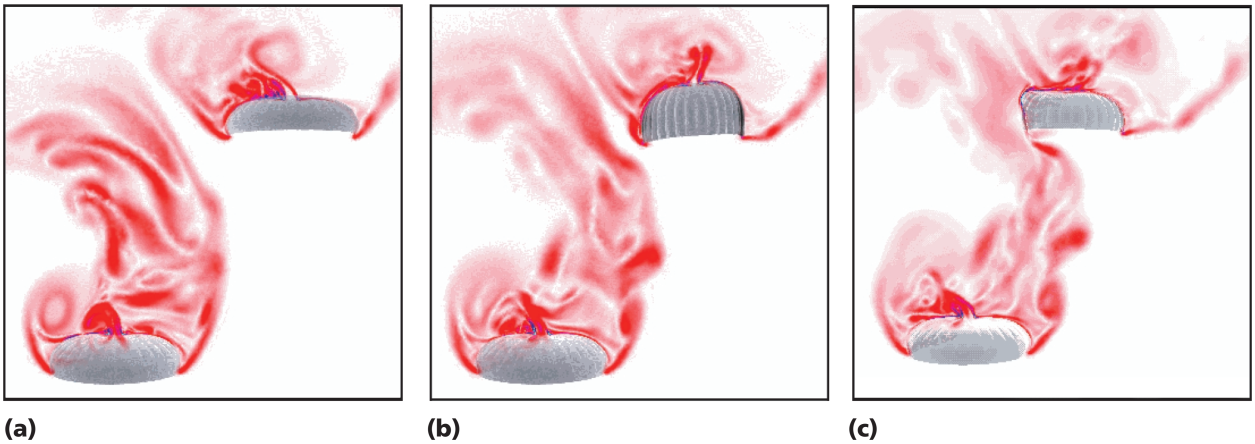

Figure 4. Changes occurring in the vorticity field surrounding two parachutes at (a) 0.00, (b) 1.75, and (c) 3.50 seconds. The upper parachute deforms as it enters the lower parachute's wake.

feet, which is approximately three times the (inflated) radii, and a vertical spacing of 56 feet. The parachute model represents a standard US Army T-10 personnel parachute. The T-10 has a 35-foot diameter, with 30 suspension lines and a 3.5-foot vent at the canopy apex. The suspension lines connect to four risers that attach to the paratrooper or payload.

We treat the lower canopy as a rigid body, while we let the upper canopy deform under the FD forces. We divide the SD model into six distinct material groups:

- A membrane group for the canopy
- three cable groups, one each for suspension lines, canopy reinforcements, and risers
- A concentrated mass
- A truss group for the payload

An interior surface in the fluid mesh represents the canopy. The typical volume mesh size is approximately 3.5 million elements, and applying the DSD/SST formulation gives us approximately 4.6 million coupled equations. We use the automatic mesh-moving method to handle canopy shape changes and occasionally remesh the fluid domain.

For the FD computations, we impose a uniform upstream boundary condition at the lower boundary of the computational domain to represent parachute descent velocities of 22 feet per second. We specify traction-free conditions at the outflow boundary, and zero normal velocity and zero shear stress conditions at the side boundaries. While we specify no-slip conditions on surfaces of the lower parachute, the specified surface velocities for the upper parachute come from the SD solution. Figure 4 shows the vorticity field surrounding the two parachutes, and Figure 5 shows the dynamics of the upper parachute structure during a 0.0 to 3.5-second time period.

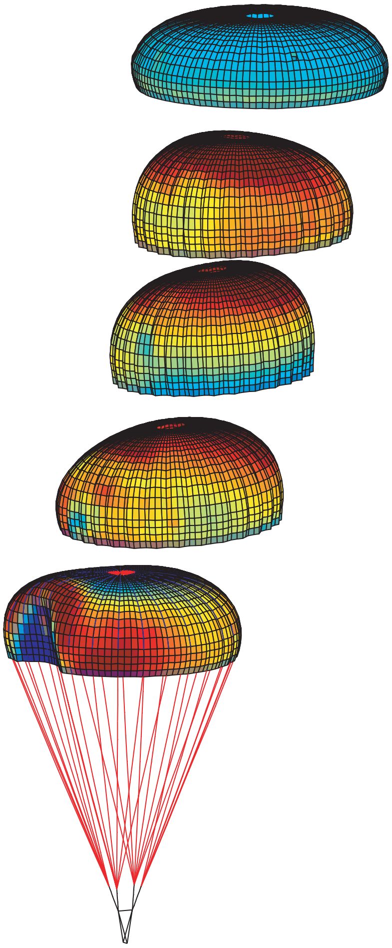

Figure 5. Deformations of the upper parachute structure in a 0.0 to 3.5-second time period. Canopy colors show differential air pressure.

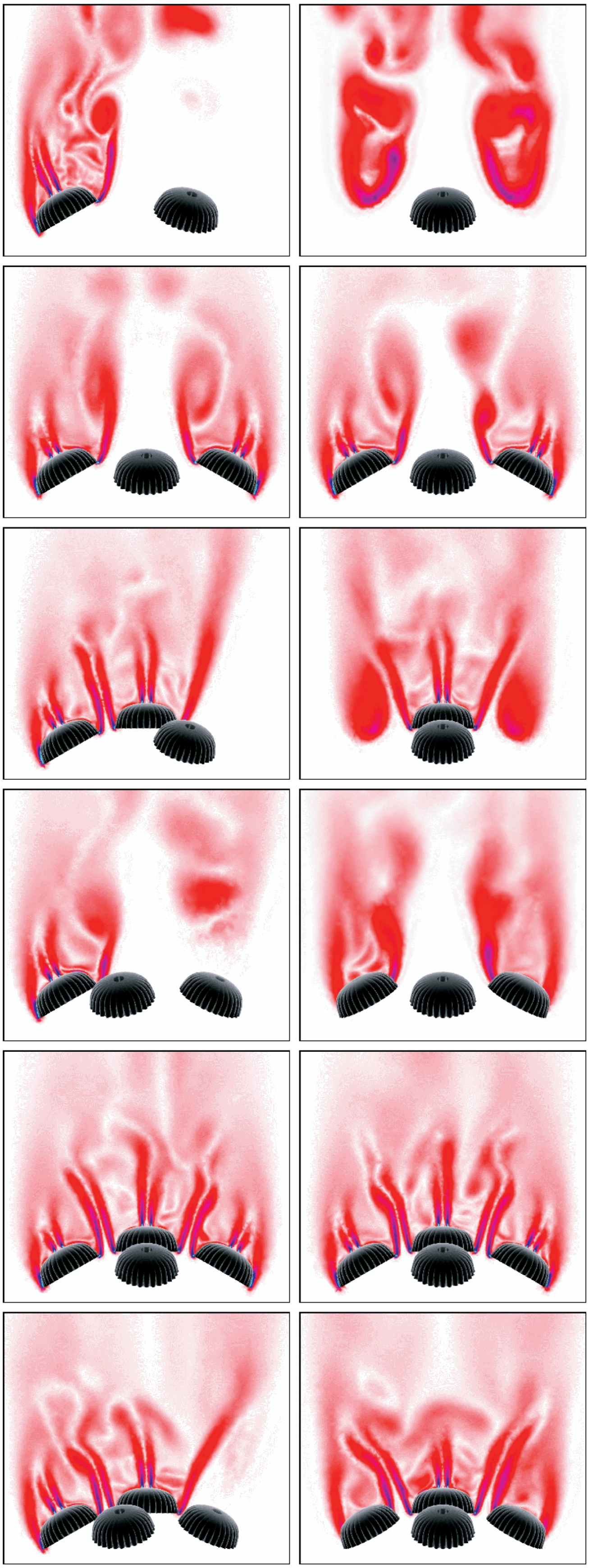

Figure 6. Interaction between canopies in a cluster of parachutes and the vorticity magnitude for two cross-sectional planes, $x = 0$ and $y = 0$. From top to bottom: three canopies, four canopies, four canopies with one at the center, five canopies, five canopies with one at the center, and six canopies with one at the center.

Between parachute clusters. We also present results for interactions between canopies in a cluster of parachutes. As with the two-parachute tests, initial simulations focused on the purely aerodynamic interactions.[10] The parachute canopies were treated as rigid geometries. We studied how varying the canopies' number (from three to six parachutes in a cluster) and arrangement influences the aerodynamic interactions between the canopies. In all the cases we studied, we had a number of canopies distributed uniformly at a prescribed angle about the azimuthal axis. In some of those cases we had a canopy also at the center. Figure 6 shows, for each arrangement, the computed flow fields from these simulations. It depicts the vorticity magnitude in two cross-sectional planes, $x = 0$ and $y = 0$. These initial simulations demonstrate the interactions between canopies in different cluster arrangements. We are currently conducting FSI simulations to address additional factors, such as the preferred arrangements and dynamics of the canopies.

Parachute Control-Line Inputs

In many parachute applications, we would want to know how the parachute would respond to control-line inputs. These control-line inputs would be performed by the paratrooper or, in cargo parachutes, by a special device. For example, in a round cargo parachute, single or multiple control lines (risers) can be extended to attain limited glide performance. The US Army's Affordable Guided Airdrop System (AGAS),[13] which is a steerable round parachute, is capable of such control-line inputs. To give another example, a paratrooper with a round parachute can pull one or two of the risers to adjust his or her orientation just before landing. We have conducted preliminary FSI simulations for these two types of control-line inputs.[12]

As a third example, we can mention the control-line inputs performed by the soft-landing retraction systems. In these applications, a control line or "muscle" between the payload and suspension lines provides rapid contraction at landing, reducing landing speed.

Steerable Round Parachutes

We are also performing simulations to predict the response of a T-10 parachute to various one- and two-riser releases and are using these simulations to study parachutes' gliding behavior and to better understand the limits of round parachute glide performance. Figure 7 shows the dynamic behavior in the case of a single, released riser. The colors on the deforming canopy represent the differential pressures from the fluid dynamics computation, with red and blue representing high and low values, respectively.

Recent years have seen major advances in the computational modeling of parachute systems. These advances were based on new developments in computer hardware (such as new and more powerful parallel computers) and state-of-the-art numerical methods (such as the new formulations for flow problems with moving boundaries and interfaces).

These advances will not only help us address some of the major computational challenges in parachute simulations, but can also help us address similar challenges in other disciplines. For example, in cardiovascular fluid mechanics, a realistic modeling tool must have a fluid-structure interaction capability, because we must account for the response of the blood vessels or the heart to the fluid dynamics forces generated by the blood flow. Our current efforts are focused on improving our fluid-structure interaction and mesh-moving techniques. Anticipated future developments will continue to expand the role that computational modeling has in developing advanced parachute systems.

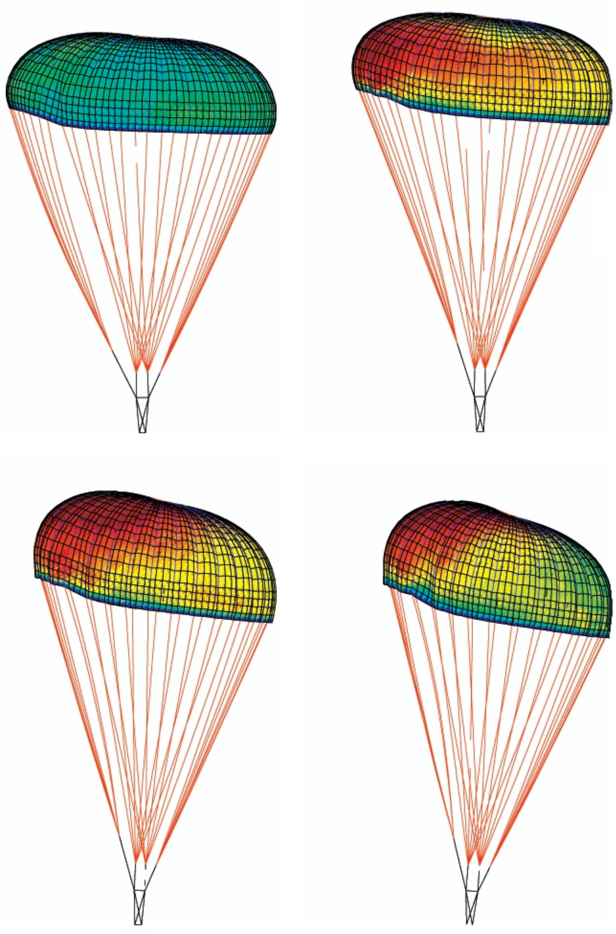

Figure 7. T-10 parachute with an extended riser. The riser release sequence is from left to right and top to bottom. The three nonextended risers have an unstressed length of 2.5 feet, and the fourth riser has an extended, unstressed length of 5.5 feet. The parachute's structure responds to the flow field and begins to pitch and glide due to the released riser.

Acknowledgments

This work was supported by the US Army Natick Soldier Center, the NASA Johnson Space Center, and the US National Science Foundation.

References

1. T.E. Tezduyar, "Stabilized Finite Element Formulations for Incompressible Flow Computations," *Advances in Applied Mechanics*, vol. 28, 1991, pp. 1–44. ← 1992 (correct publication year)
2. M.L. Accorsi et al., "Structural Modeling of Parachute Dynamics," *AIAA J.*, vol. 38, 2000, pp. 139–146.
3. T.E. Tezduyar, "Finite Element Methods for Flow Problems with Moving Boundaries and Interfaces," *Archives of Computational Methods in Eng.*, vol. 8, 2001, pp. 83–130.
4. T.J.R. Hughes and A.N. Brooks, "A Multidimensional Upwind Scheme with no Crosswind Diffusion," *Finite Element Methods for Convection Dominated Flows*, T.J.R. Hughes, ed., Applied Mechanics Division (AMD) vol. 34, Am. Soc. of Mechanical Engineers, 1979, pp. 19–35.
5. T.E. Tezduyar et al., "Computation of Unsteady Incompressible Flows with the Finite Element Methods—Space–Time Formulations, Iterative Strategies, and Massively Parallel Implementations," *New Methods in Transient Analysis*, AMD vol. 143, Am. Soc. of Mechanical Engineers, 1992, pp. 7–24.
6. K. Stein, T. Tezduyar, and R. Benney, "Mesh Moving Techniques for Fluid–Structure Interactions with Large Displacements," ~~to be published in~~ *J. Applied Mechanics*, ~~2003.~~ **70** (2003) 58-63.
7. T.E. Tezduyar, "Finite Element Interface-Tracking and Interface-Capturing Techniques for Flows with Moving Boundaries and Interfaces," *Proc. ASME Symp. Fluid-Physics and Heat Transfer for Macro- and Micro-Scale Gas-Liquid and Phase-Change Flows*, Am. Soc. of Mechanical Engineers, 2001.
8. T.E. Tezduyar, "Stabilized Finite Element Formulations and Interface-Tracking and Interface-Capturing Techniques for Incompressible Flows," to be published in *Proc. Workshop on Numerical Simulations of Incompressible Flows,* 2002. ← See update U1 below.
9. K. Stein and T. Tezduyar, "Advanced Mesh Update Techniques for Problems Involving Large Displacements," *Proc. 5th World Congress on Computational Mechanics*, Vienna Univ. of Technology, Austria; http://wccm.tuwien.ac.at/paperID 81489.

U1. T.E. Tezduyar, "Stabilized Finite Element Formulations and Interface-Tracking and Interface-Capturing Techniques for Incompressible Flows", *Numerical Simulations of Incompressible Flows* (ed. M.M. Hafez), World Scientific, New Jersey (2003) 221-239.

10. K. Stein et al., "Aerodynamic Interaction between Multiple Parachute Canopies," *Proc. 1st MIT Conf. Computational Fluid and Solid Mechanics*, CD-ROM, Elsevier, 2001, pp. 968–972.

See update U2 below.

11. K. Stein et al., "Computational Modeling of Parachute Fluid-Structure Interactions," to be published in *Proc. Workshop on Numerical Simulations of Incompressible Flows*, 2002.

12. K. Stein, T. Tezduyar, and R. Benney, "Applications in Airdrop Systems: Fluid-Structure Interaction Modeling," *Proc. 5th World Congress on Computational Mechanics*, Vienna Univ. of Technology, Austria, 2002; http://wccm.tuwien.ac.at/paperID81545.

13. R. Almassy et al., "The Affordable Guided Airdrop System (AGAS)," *Proc. CEAS/AIAA 15th Aerodynamic Decelerator Systems Technology Conf.*, Am. Inst. of Aeronautics and Astronautics, paper no. AIAA-99-1742, 1999, pp. 316–325.

U2. K. Stein, T. Tezduyar, R. Benney, M. Accorsi and H. Johari, "Computational Modeling of Parachute Fluid-Structure Interactions", *Computational Fluid Dynamics Journal*, **12** (2003) 516-526.

Keith Stein is an associate professor of physics at Bethel College and an adjunct associate professor in mechanical engineering at Rice University. His research interests are in fluid-structure interactions and parachute aerodynamics. He received a BA in physics from Bethel College and an MS in aerospace engineering from the University of Minnesota. As an Army researcher at the US Army Soldier Systems Center, he received a PhD in aerospace engineering from the University of Minnesota. He is the member of the American Institute of Aeronautics and Astronautics, the American Society of Mechanical Engineers, the International Association for Computational Mechanics, and the American Association of Physics Teachers. Contact him at Bethel College, 3900 Bethel Dr., St. Paul, MN 55112; k-stein@bethel.edu.

Tayfun Tezduyar is the James F. Barbour Professor of Mechanical Engineering at Rice University. His research interests include flow simulation and modeling, computational mechanics, fluid–structure interactions, and parachute simulations. He received a PhD in mechanical engineering from the California Institute of Technology. He is a fellow of the American Society of Mechanical Engineers, the International Association for Computational Mechanics, and the United States Association for Computational Mechanics. Contact him at Dept. of Mechanical Eng., Rice Univ. - MS 321, 6100 Main St., Houston, TX 77005; tezduyar@rice.edu.

Richard Benney is an aerospace engineer and team leader of the Airdrop Technology Team at the US Army Soldier Systems Center. His research interests are in simulation and modeling of parachute and other airdrop systems. He received his BS and MS in mechanical engineering from Northeastern University and an MS in applied sciences from Harvard. He is an active member of the American Institute of Aeronautics and Astronautics. Contact him at the US Army Soldier and Biological Chemical Command, Natick Soldier Center, ATTN: AMSSB-RAD-T(N), Kansas St., Natick, MA 01760-5017; Richard.Benney@natick.army.mil.