Aerodynamics of the Crew Return Vehicle and parafoil at different opening stages

Lea Haubelt, Rachel Melton, Weiqing Yee, Tayfun Tezduyar *

Team for Advanced Flow Simulation and Modeling (T*AFSM)1, Mechanical Engineering and Materials Science, Rice University, MS 321, 6100 Main Street, Houston, TX 77005, USA

Abstract

We present computation of the aerodynamics of the Crew Return Vehicle of the International Space Station and the ram-air parachute (parafoil) to be used with it. Deceleration, gliding and landing of the CRV will involve different types of parachutes and various stages, and the parafoil will have six different opening stages. Of these six stages, we focus our attention to four that are symmetric in parafoil geometry. The airspeed during the parafoil operation will be low enough to reasonably assume that the unsteady flow to be computed is governed by the Navier–Stokes equations of incompressible flows. The computational method is based on a stabilized finite element formulation with the Streamline-Upwind/Petrov–Galerkin and Pressure-Stabilizing/Petrov–Galerkin stabilizations. The computations are carried out on a distributed-memory parallel supercomputer.

Keywords: International space station; Crew return vehicle; Soft landing; Parachute; Parafoil; Opening stages

1. Introduction

Normally, the Space Shuttle will transport the Crew to and from the Space Station. However, in some cases, it might be necessary to bring the Crew back sooner than the time required to send the Shuttle up. In such cases, the Crew will use the Crew Return Vehicle (CRV), which can quickly be launched from the Space Station and can re-enter the atmosphere. However, the CRV, which is small and of fixed configuration, will only utilize a sequence of parachutes to first slow it down, then glide it to the landing area, and finally, land it softly. Different types of parachutes will be used to accomplish these goals, starting with round parachutes to slow the CRV down, and ending with a large-ram air parachute (parafoil) to glide and land it (see Fig. 1). Using parafolies to glide and soft-land large objects like the CRV is a new concept, and it is difficult to perform laboratory tests with parafolies of the sizes required to land such large objects. The purpose of the research presented in this paper is to develop a set of computational methods to simulate this class of problems.

The parafoil will go through a number of opening stages, starting with the smallest size in the first stage, expanding to its full size in the subsequent stages, and

Fig. 1. Crew return vehicle and parafoil in a flight test.

* Corresponding author. Tel.: +1 (713) 348-6051; Fax: +1 (713) 348-5423; E-mail: tezduyar@rice.edu
1 http://www.mems.rice.edu/TAFSM/
reaching its final geometric shape in the last stage. Simulation of progressing through these stages involves a number of computational challenges, including time-dependent shape transformations and fluid–structure interactions. Although we addressed these types of challenges in the past in the context of other applications (see [1,2]), here we focus on computation of the discrete stages of the parafoil, without taking into account the fluid–structure interactions.

It is assumed that the airspeed during different disreefing stages of the parafoil is not high enough for the compressibility effects to be significant. Consequently, we use the Navier–Stokes equations of incompressible flows as the governing equations for the airflow. These equations are solved with a semi-discrete finite element formulation, based on the Streamline-Upwind/Petrov–Galerkin (SUPG) [3,4] and Pressure-Stabilizing/Petrov–Galerkin (PSPG) [5] stabilizations. These formulations remain stable in computation of flows with high Reynolds numbers and boundary layers, without introducing excessive numerical dissipation. They also allow us to use, without facing numerical instability problems, equal-order interpolation functions for velocity and pressure. This is a useful feature in implementing these methods for efficient parallel computations.

The class of problems we want to solve results in large, coupled nonlinear equation systems that need to be solved at every time step. We solve these equations with the Newton–Raphson method. At each step of the Newton–Raphson sequence, we are faced with solving a large, coupled linear equation system. This linear system be solved at every time step. We solve these equations with the Newton–Raphson method. At each step of the Newton–Raphson sequence, we are faced with solving a linear system.

2. Governing equations

Let \( \Omega, \subset \mathbb{R}^d \) 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 as

\[
\rho \frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} - \mathbf{f} = \nabla \cdot \mathbf{\sigma} = 0 \quad \text{on} \quad \Omega, \quad \forall t \in (0, T),
\]

\[
\nabla \cdot \mathbf{u} = 0 \quad \text{on} \quad \Omega, \quad \forall t \in (0, T),
\]

where \( \rho \), \( \mathbf{u} \) and \( \mathbf{f} \) are the density, velocity and the external force, respectively. The stress tensor, \( \mathbf{\sigma}(\rho, \mathbf{u}) = -\rho \mathbf{I} + 2\mu \mathbf{e}^{\mathbf{u}} \), is given. Here, \( \rho \) and \( \mathbf{I} \) are the pressure, identity tensor and the viscosity. The strain rate tensor, \( \mathbf{e}^{\mathbf{u}} = \frac{1}{2} \left( \nabla \mathbf{u} + (\nabla \mathbf{u})^T \right) \), both Dirichlet- and Neumann-type boundary conditions are accounted for: \( \mathbf{u} = \mathbf{g} \) on \( \Gamma_{\ell} \) and \( \mathbf{n} \cdot \mathbf{u} = \mathbf{h} \) on \( \Gamma_n \). Here \( \Gamma_{\ell} \) and \( \Gamma_n \) are complementary subsets of \( \Gamma \), \( \mathbf{n} \) is the unit normal vector at the boundary, and \( \mathbf{g} \) and \( \mathbf{h} \) are given. A divergence-free velocity field is specified as the initial condition.

3. Finite element formulation

Let us consider a fixed spatial domain \( \Omega \) and its boundary \( \Gamma \), where subscript \( t \) is dropped from both \( \Omega \) and \( \Gamma_t \). The domain \( \Omega \) is discretized into sub-domains \( \Omega^e \), \( e = 1, \ldots, n_e \), where \( n_e \) is the number of elements. For this discretization, the finite element trial functions \( S^h \) for velocity and \( S^h \) for pressure, and the corresponding test function spaces \( Y^h \) and \( Y^h \) are defined as follows:

\[
S^h = \{ \mathbf{u}^h | \mathbf{u}^h \in [H^1(\Omega)]^d, \mathbf{u}^h \equiv g^h \text{ on } \Gamma_g \}, \]

\[
Y^h = \{ \mathbf{w}^h | \mathbf{w}^h \in [H^1(\Omega)]^d, \mathbf{w}^h \equiv 0 \text{ on } \Gamma_g \},
\]

\[
S^h = \{ q^h | q^h \in H^1(\Omega) \}.
\]

Here \( H^1(\Omega) \) is the finite-dimensional function space over \( \Omega \). The stabilized finite element formulation is written as follows: find \( \mathbf{u}^h \in S^h \) and \( p^h \in S^h \) such that \( \forall \mathbf{w}^h \in Y^h \) and \( q^h \in Y^h \):

\[
\int_\Omega \mathbf{w}^h : \rho \left( \frac{\partial \mathbf{u}^h}{\partial t} + \mathbf{u}^h \cdot \nabla \mathbf{u}^h - \mathbf{f}^h \right) \, d\Omega \\
+ \int_\Omega \mathbf{e}^{\mathbf{u}^h} : \mathbf{\sigma}(p^h, \mathbf{u}^h) \, d\Omega + \int_\Omega q^h \nabla \cdot \mathbf{u}^h \, d\Omega \\
+ \sum_{e=1}^{n_e} \int_{\Omega^e} \frac{1}{\rho} \left[ \tau_{\text{SISC}} \mathbf{u}^h \cdot \nabla \mathbf{w}^h + \tau_{\text{SISC}} \nabla q^h \right] \\
- \left[ \rho \left( \frac{\partial \mathbf{u}^h}{\partial t} + \mathbf{u}^h \cdot \nabla \mathbf{u}^h \right) - \nabla \cdot \mathbf{\sigma}(p^h, \mathbf{u}^h) - \rho \mathbf{f}^h \right] \, d\Omega \\
+ \sum_{e=1}^{n_e} \int_{\Gamma_{\ell}} \tau_{\text{SISC}} \nabla \cdot \mathbf{w}^h \rho \nabla \cdot \mathbf{u}^h d\Omega = \int_{\Gamma_{\ell}} \mathbf{w}^h \cdot \mathbf{h}^h d\Gamma.
\]

In this formulation, \( \tau_{\text{SISC}}, \tau_{\text{SPG}}, \) and \( \tau_{\text{LSC}} \) are the stabilization parameters (see [6]). For an earlier, detailed reference on this stabilized formulation see [5].

4. Numerical simulations

In our simulations, the CRV model has a length of 24.2 ft and is descending at 50 ft/s. The parafoil goes through six different opening stages, and we focus on the four that are symmetric in parafoil geometry. Computations are carried out by taking each of these four stages as a separate case. Fig. 2 shows the first three symmetric opening stages of the parafoil. In Stage 1, shown in the top picture, the parafoil has 11 fully disreefed cells, and the span length is 38.5 ft. Stage 2, shown in the middle picture, has 23 disreefed cells and a span length of 80.5 ft. The last picture
Fig. 2. First three symmetric stages of the parafoil. Top: Stage 1 (11 disreefed cells, span length $= 38.5$ ft). Middle: Stage 2 (23 disreefed cells, span length $= 80.5$ ft). Bottom: Stage 3 (31 disreefed cells, span length $= 108.5$ ft).

Fig. 3. Final stage of the parafoil is reached by increasing the radius of curvature of Stage 3. Top: Stage 3 prior to the increase. Bottom: after the increase.

in this figure shows Stage 3, which has 31 disreefed cells and a span length of 108.5 ft. In the final stage, Stage 4, the parafoil has the same number of disreefed cells and span length as it did in Stage 3, but an increased radius of curvature. Fig. 3 shows how the final stage is reached from Stage 3 by increasing the radius of curvature. The number of elements in the tetrahedral meshes used for the different stages of the parafoil ranges from 2.2 million to 3.5 million. Fig. 4 shows, for the final stage of the parafoil, the air pressure distribution on the surface of the CRV and parafoil.

5. Concluding remarks

We have presented the initial phases of our computational work in simulation of the aerodynamics of the Crew Return Vehicle of the International Space Station and the parafoil to be used for its gliding and soft-landing. The parafoil has six different opening stages, and we focused our attention to four that are symmetric. The computations were based on taking each of these four stages as a separate case, and obtaining time-dependent solution for each. The method used is a stabilized, semi-discrete finite element formulation with the SUPG and PSPG stabilizations, and the parallel computations were carried out on a CRAY T3E-1200.

Acknowledgements

The work reported in this paper was partially sponsored by NASA JSC (Grant NAG9-1059) and by the Natick Soldier Center (Contract DAAD16-00-C-9222). The content does not necessarily reflect the position or the policy of the government, and no official endorsement should be inferred.
References


