Fully-Coupled Fluid-Structure Interaction Simulations of a Supersonic Parachute

Jonathan Boustani†*, Michael F. Barad†, Cetin C. Kiris†
† Computational Aerosciences Branch, NASA Ames Research Center, Moffett Field, CA, 94035, USA
Christoph Brehm∗
∗Department of Mechanical Engineering, University of Kentucky, Lexington, KY, 40506, USA

Advanced Modeling & Simulation (AMS) Seminar Series
NASA Ames Research Center Thursday, September 19th 2019
Outline

- Motivation/Introduction
  - Mars, EDL system qualification, Simulation Capabilities

- FSI Method
  - Governing equations, Overview
  - Immersed Boundary Method for the Compressible Navier-Stokes Equations (CFD)
  - Geometrically Nonlinear Computational Structural Dynamics Solver (CSD)
  - Coupling procedure

- Summary of FSI Method Validation

- Methods for Large-scale, Parallel CFD-CSD Coupling
  - Disparate domain decomposition

- Supersonic Parachute Inflation

- Summary and Outlook
Outline

- **Motivation/Introduction**
  - Mars, EDL system qualification, Simulation Capabilities

- **FSI Method**
  - Governing equations, Overview
  - Immersed Boundary Method for the Compressible Navier-Stokes Equations (CFD)
  - Geometrically Nonlinear Computational Structural Dynamics Solver (CSD)
  - Coupling procedure

- **Summary of FSI Method Validation**

- **Methods for Large-scale, Parallel CFD-CSD Coupling**
  - Disparate domain decomposition

- **Supersonic Parachute Inflation**

- **Summary and Outlook**
MSL EDL system was requalified
  - Payload weight, canopy size, and landing altitude exceeded those established by Viking mission

LDSD missions
  - Successful wind tunnel tests, parachute failure in real world tests

NASA’s mission to Mars will eventually require EDL re-qualification again
  - For hardware and humans required for sustained settlements, more demanding landing objectives
Introduced and validated a method for simulating the large, geometrically nonlinear deformations of very thin shell structures (Boustani et al. SciTech 2019)

Extended these capabilities to solving large-scale FSI problems in high-speed flows within a parallel computing environment

End goal is high-fidelity supersonic parachute deployment simulations
**LAVA Framework**

- **Structured Cartesian AMR Navier-Stokes**
- **Unstructured Arbitrary Polyhedral Navier-Stokes**
- **Structured Curvilinear Navier-Stokes**
- **Post-Processing Tools**
- **Far Field Acoustic Solver**
- **Conjugate Heat Transfer**
- **Actuator Disk Models**
- **Structural Dynamics**
- **6 DOF Body Motion**

**LAVA**
Object Oriented Framework
C++ / Fortran with MPI Parallel
Domain Connectivity/ Shared Data

- **Space-Marching Propagation**
- **Other Solvers & Frameworks**

**Multi-Physics:**
- Multi-Phase Combustion Chemistry
- Electro-Magnetics

**Other Development Efforts**
- Higher order and low dissipation
- Curvilinear grid generation
- Wall modeling
- LES/DES/ILES Turbulence
- HEC (optimizations, accelerators, etc)

---

*Kiris at al. AIAA-2014-0070 & AST-2016*
LAVA Framework

- Structured Cartesian AMR Navier-Stokes
- Lattice Boltzmann
- Post-Processing Tools
- Far Field Acoustic Solver
- Conjugate Heat Transfer
- Actuator Disk Models
- Structural Dynamics
- 6 DOF Body Motion
- Space-Marching Propagation
- Other Solvers & Frameworks
- Multi-Physics: Multi-Phase Combustion Chemistry Electro-Magnetics

LAVA
Object Oriented Framework
C++ / Fortran with MPI Parallel
Domain Connectivity/ Shared Data

Other Development Efforts
- Higher order and low dissipation
- Curvilinear grid generation
- Wall modeling
- LES/DES/ILES Turbulence
- HEC (optimizations, accelerators, etc)

Kiris at al. AIAA-2014-0070 & AST-2016
Computational Grid Paradigms in LAVA

<table>
<thead>
<tr>
<th>Structured Cartesian AMR</th>
<th>Unstructured Arbitrary Polyhedral</th>
<th>Structured Curvilinear</th>
</tr>
</thead>
<tbody>
<tr>
<td>• Essentially no manual grid generation</td>
<td>• Partially automated grid generation</td>
<td>• High quality body fitted grids</td>
</tr>
<tr>
<td>• Highly efficient Structured Adaptive Mesh Refinement (AMR)</td>
<td>• Body fitted grids</td>
<td>• Low computational cost</td>
</tr>
<tr>
<td>• Low computational cost</td>
<td>• Grid quality can be challenging</td>
<td>• Reliable higher order methods</td>
</tr>
<tr>
<td>• Reliable higher order methods</td>
<td>• High computational cost</td>
<td>• Higher order methods yet to fully mature</td>
</tr>
<tr>
<td>• Non-body fitted -&gt; Resolution of boundary layers inefficient</td>
<td></td>
<td>• Grid generation largely manual and time consuming</td>
</tr>
</tbody>
</table>
## Computational Grid Paradigms in LAVA

<table>
<thead>
<tr>
<th>Structured Cartesian AMR</th>
<th>Unstructured Arbitrary Polyhedral</th>
<th>Structured Curvilinear</th>
</tr>
</thead>
<tbody>
<tr>
<td>- Essentially no manual grid generation</td>
<td></td>
<td></td>
</tr>
<tr>
<td>- High efficiency Structured Adaptive Mesh Refinement (AMR)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>- Low computational cost</td>
<td></td>
<td></td>
</tr>
<tr>
<td>- Reliable higher order methods</td>
<td></td>
<td></td>
</tr>
<tr>
<td>- Non-body fitted -&gt; Resolution of boundary layers inefficient</td>
<td></td>
<td></td>
</tr>
<tr>
<td>- Partially automated grid generation</td>
<td></td>
<td></td>
</tr>
<tr>
<td>- Body fitted grids</td>
<td></td>
<td></td>
</tr>
<tr>
<td>- Grid quality can be challenging</td>
<td></td>
<td></td>
</tr>
<tr>
<td>- High computational cost</td>
<td></td>
<td></td>
</tr>
<tr>
<td>- Higher order methods yet to fully mature</td>
<td></td>
<td></td>
</tr>
<tr>
<td>- High quality body fitted grids</td>
<td></td>
<td></td>
</tr>
<tr>
<td>- Low computational cost</td>
<td></td>
<td></td>
</tr>
<tr>
<td>- Reliable higher order methods</td>
<td></td>
<td></td>
</tr>
<tr>
<td>- Grid generation largely manual and time consuming</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Motivation/Introduction
- Mars, EDL system qualification, Simulation Capabilities

FSI Method
- Governing equations, Overview
  - Immersed Boundary Method for the Compressible Navier-Stokes Equations (CFD)
  - Geometrically Nonlinear Computational Structural Dynamics Solver (CSD)
  - Coupling procedure

Summary of FSI Method Validation

Methods for Large-scale, Parallel CFD-CSD Coupling
- Disparate domain decomposition

Supersonic Parachute Inflation

Summary and Outlook
The fluid regime considers the compressible Navier-Stokes equations, shown here in conservative form

\[
\frac{\partial \mathbf{W}}{\partial t} + \frac{\partial \mathbf{E}}{\partial x} + \frac{\partial \mathbf{F}}{\partial y} + \frac{\partial \mathbf{G}}{\partial z} = 0
\]

\[
\mathbf{W} = \begin{bmatrix}
\rho, \rho u, \rho v, \rho w, \rho e_t
\end{bmatrix}^T
\]

The structural regime considers the Total Lagrangian equations of motion

\[
\int_{V_0}^V S_{ij} \delta_0 \epsilon_{ij} \, d^0 V + \int_{V_0}^V S_{ij} \delta_0 \eta_{ij} \, d^0 V = (t + \Delta t) \mathcal{R} - \int_{V_0}^V S_{ij} \delta_0 e_{ij} \, d^0 V
\]

Partitioned solution involves solving strong and weak solutions together
The method used in this work couples together

I. A block-structured Cartesian, higher-order, sharp, immersed boundary method for the compressible Navier-Stokes equations

   Brehm, C., Fasel, H., *JCP* 2013
   Kiris et al., *Aerospace Sci. and Tech* 2016
   Brehm, C., Barad, M. F., Kiris, C. C., *JCP* 2019

II. A geometrically nonlinear structural finite element solver employing shell elements that utilize the Mixed Interpolation of Tensorial Components

   Boustani et al., *AIAA SciTech* 2019
   Boustani et al., *AIAA Aviation* 2019
Motivation/Introduction
  - Mars, EDL system qualification, Simulation Capabilities

FSI Method
  - Governing equations, Overview
  - Immersed Boundary Method for the Compressible Navier-Stokes Equations (CFD)
  - Geometrically Nonlinear Computational Structural Dynamics Solver (CSD)
  - Coupling procedure

Summary of FSI Method Validation

Methods for Large-scale, Parallel CFD-CSD Coupling
  - Disparate domain decomposition

Supersonic Parachute Inflation

Summary and Outlook
Higher-Order IBM

- **Stability is built into derivation**
  - In contrast to proving *a posteriori*

- Automatic volume mesh generation
  - For any arbitrarily complex geometry

- FD stencil coefficients at irregular grid points are locally optimized
  - Improved global stability
Higher-Order IBM

- Stability is built into derivation
  - In contrast to proving *a posteriori*

- **Automatic volume mesh generation**
  - For any arbitrarily complex geometry

- FD stencil coefficients at irregular grid points are locally optimized
  - Improved global stability
Higher-Order IBM

- Stability is built into derivation
  - In contrast to proving *a posteriori*

- Automatic volume mesh generation
  - For any arbitrarily complex geometry

- FD stencil coefficients at irregular grid points are locally optimized
  - Improved global stability
Higher-Order IBM

- Stability is built into derivation
  - In contrast to proving *a posteriori*

- Automatic volume mesh generation
  - For any arbitrarily complex geometry

- ‘Sharp’ classification → BCs applied at intersection points
  - No valid data needed inside geometry
  - In other words, no cells needed inside geometry
Need to deal with freshly-cleared cells (FCCs)

I. Inside/outside geometry (●)
II. Flipped boundary distance (▲)

No valid time-history → extrapolate

To reduce spurious oscillations in the pressure field, the stencil should be chosen to:
- Extend into the direction with the maximum body velocity (low speed flows)
- Utilize ENO (in supersonic flows)
Higher-Order IBM

- Solution is obtained with:
  1. 2\textsuperscript{nd}-/4\textsuperscript{th}-order time integration (RK)
  2. 5\textsuperscript{th}-order spatial discretization (WENO)

- Framework can handle geometries much smaller than the minimum grid spacing

- Robust extrapolation of flow field quantities onto the surface
Higher-Order IBM

- Developing methods for robust extrapolating of flow quantities onto very thin and complex geometries
This ensures smooth pressure and viscous loading for the CSD solver.
This ensures smooth pressure and viscous loading for the CSD solver.
This ensures smooth pressure and viscous loading for the CSD solver.
Outline

- **Motivation/Introduction**
  - Mars, EDL system qualification, Simulation Capabilities

- **FSI Method**
  - Governing equations, Overview
  - Immersed Boundary Method for the Compressible Navier-Stokes Equations (CFD)
  - Geometrically Nonlinear Computational Structural Dynamics Solver (CSD)
  - Coupling procedure

- **Summary of FSI Method Validation**

- **Methods for Large-scale, Parallel CFD-CSD Coupling**
  - Disparate domain decomposition

- **Supersonic Parachute Inflation**

- **Summary and Outlook**
Element formulations used:
- Geometrically nonlinear MITC3 triangular shell element
  \((\text{Lee and Bathe, } \textit{Comp. and Struc. } 2004)\)
- 2-node geometrically nonlinear cable element

MITC3 shell element mitigates ‘shear locking’ via an assumed shear strain field:
\[
\tilde{e}_{rt} = e_{rt}^{(1)} + cs \\
\tilde{e}_{st} = e_{st}^{(2)} - cr \\
c = e_{st}^{(2)} - e_{rt}^{(1)} - e_{st}^{(3)} + e_{rt}^{(3)}
\]

Substitute displacement-based shear strain components for assumed shear strains
Element formulations used:
- Geometrically nonlinear MITC3 triangular shell element
  *(Lee and Bathe, *Comp. and Struc.* 2004)*
- 2-node geometrically nonlinear cable element

St. Venant-Kirchoff hyperelastic strain-energy function

Time integration is performed with the implicit Newmark-β scheme
- Dynamic Newton-Raphson relaxation parameter optimizes convergence properties
- Can converge very thin shell structures in steady and unsteady simulations
- Robust to coarse CFD resolution loading and acoustic disturbances
CSD solver validated for canonical large deformation problems

(Boustani et al., AIAA 2019)
CSD solver validated for canonical large deformation problems
(Boustani et al., AIAA 2019)
Outline

- **Motivation/Introduction**
  - Mars, EDL system qualification, Simulation Capabilities

- **FSI Method**
  - Governing equations, Overview
  - Immersed Boundary Method for the Compressible Navier-Stokes Equations (CFD)
  - Geometrically Nonlinear Computational Structural Dynamics Solver (CSD)
  - Coupling procedure

- **Summary of FSI Method Validation**

- **Methods for Large-scale, Parallel CFD-CSD Coupling**
  - Disparate domain decomposition

- **Supersonic Parachute Inflation**

- **Summary and Outlook**
The coupling conditions between the two regimes enforce the continuity of loads across the shared boundary

\[ t_{\text{structure}}(\overline{x}_b(t), t) = t_{\text{fluid}}(\overline{x}_b(t), t) \]

where the fluid traction vector considers pressure and viscous stresses.

The continuity of the position and velocity of the shared boundary itself is also enforced

\[ \overline{x}_b(t) = x_{\text{fluid}}(t) = x_{\text{structure}}(t), \text{ and } \]
\[ \overline{\dot{x}}_b(t) = \dot{x}_{\text{fluid}}(t) = \dot{x}_{\text{structure}}(t), \forall t \geq 0 \]
The CFD and CSD solvers are weakly coupled
- The solution procedure is **partitioned**
- Parachute canopy is 33,000x more dense than upper Martian atmosphere

An auxiliary and mass-less, or phantom, representation of the geometry with a finite thickness is used in the CFD solver

The coupling conditions are enforced on the geometry representation
The CFD and CSD solvers are weakly coupled
- The solution procedure is **partitioned**
- Parachute canopy is 33,000x more dense than upper Martian atmosphere

An auxiliary and mass-less, or phantom, representation of the geometry with a finite thickness is used in the CFD solver

The coupling conditions are enforced on the geometry representation
Outline

- **Motivation/Introduction**
  - Mars, EDL system qualification, Simulation Capabilities

- **FSI Method**
  - Governing equations, Overview
  - Immersed Boundary Method for the Compressible Navier-Stokes Equations (CFD)
  - Geometrically Nonlinear Computational Structural Dynamics Solver (CSD)
  - Coupling procedure

- **Summary of FSI Method Validation**

- **Methods for Large-scale, Parallel CFD-CSD Coupling**
  - Disparate domain decomposition

- **Supersonic Parachute Inflation**

- **Summary and Outlook**

Parabolic velocity profile at the inlet

\[ u(0, y) = 1.5\bar{U} \frac{y(H - y)}{(H/2)^2} = 1.5\bar{U} \frac{4.0}{0.1681} y(0.41 - y) \]

Normalized excursion amplitude and Strouhal number are compared
Case 1:
- $Re = 100$
- $DR = 10$
- $S_b = \frac{E_s}{\rho_f U^2} = 1,400$
- $\Delta x_{min} = \Delta y_{min} = 0.16D$
- 100 beam elements, 100 blocks of triangular elements

Color contours of vertical velocity
Fixed Cylinder with Trailing Filament

Results from Geometrically Nonlinear FSI Simulation

<table>
<thead>
<tr>
<th>Reference</th>
<th>$\bar{A}$</th>
<th>$St$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Current Research</td>
<td>0.84</td>
<td>0.18</td>
</tr>
<tr>
<td>Turek and Hron (2006)</td>
<td>0.83</td>
<td>0.19</td>
</tr>
<tr>
<td>Tian et al. (JCP 2013)</td>
<td>0.78</td>
<td>0.19</td>
</tr>
<tr>
<td>Bhardwaj and Mittal (AIAA Journal 2012)</td>
<td>0.92</td>
<td>0.19</td>
</tr>
</tbody>
</table>

$$\bar{A} = \frac{1}{2} \frac{d_{y,max} - d_{y,min}}{D}$$

$$St = f \frac{D}{U}$$
Case 2:

- \(Re = 200\)
- \(DR = 1\)
- \(S_b = \frac{E_s}{\rho_f U^2} = 1,400\)
- \(\Delta x_{min} = \Delta y_{min} = 0.16D\)
- 100 beam elements, 100 blocks of triangular elements

\[
\bar{A} = \frac{1}{2} \frac{d_{y,max} - d_{y,min}}{D}
\]

\[
St = f \frac{D}{U}
\]

### Results from Geometrically Nonlinear FSI Simulation

<table>
<thead>
<tr>
<th>Reference</th>
<th>(\bar{A})</th>
<th>(St)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Current Research</td>
<td>0.29</td>
<td>0.26</td>
</tr>
<tr>
<td>Turek and Hron (2006)</td>
<td>0.36</td>
<td>0.29</td>
</tr>
<tr>
<td>Tian et al. (JCP 2013)</td>
<td>0.32</td>
<td>0.26</td>
</tr>
<tr>
<td>Bhardwaj and Mittal (AIAA Journal 2012)</td>
<td>0.41</td>
<td>0.28</td>
</tr>
</tbody>
</table>
Consider the crumpling and wrinkling of a thin, elastic sheet

- \( Re = 100 \)
- \( MR = 1 \)
- \( S_b \) and \( S_S \) vary
- \( \Delta x_{\text{min}} = \Delta y_{\text{min}} = 0.015D \)
- 3,300 finite elements discretize the disk
- Clamped core size \( d_c = 0.1 \)
Compliant Disk in Viscous Flow

\[ S_b = 10^{-1} \]
\[ S_s = 10^3 \]
Compliant Disk in Viscous Flow

\[ S_b = 4 \times 10^{-2} \]
\[ S_s = 10^3 \]
Compliant Disk in Viscous Flow

Current Research

Simulations by Hua et al. (Adjusted from JFM 2014)

Experiments by Schouveiler and Eloy (Adjusted from Physical Review Letters 2013)
Compliant Disk in Viscous Flow

\[ S_b = 10^{-3} \]
\[ S_s = 10^2 \]
Compliant Disk in Viscous Flow

M3 Comparison

Current Research

Simulations by Hua et al. 
*(Adjusted from JFM 2014)*

Experiments by Schouveiler and Eloy 
*(Adjusted from Physical Review Letters 2013)*
Compliant Disk in Viscous Flow

\[ S_b = 2 \times 10^{-4} \]
\[ S_s = 10^3 \]
Compliant Disk in Viscous Flow

$S_b = 3 \times 10^{-5}$

$S_s = 2 \times 10^2$
Compliant Disk in Viscous Flow

Color contours of vorticity magnitude

M5
Compliant Disk in Viscous Flow

\[ S_b = 2 \times 10^{-5} \]
\[ S_s = 2 \times 10^2 \]
For comparison with Hu and Wang (JAFM 2016) and Womack and Seidel (AIAA 2014), Siefers et al. (AIAA 2018) introduced the

1. Mean chord angle
\[ \phi = \tan^{-1}\left(\frac{\delta_x}{H - \delta_z}\right) \]

2. Normalized curvature
\[ k = \frac{qH^3}{Eh^3} \]

These parameters reduce the solution to a single variable, \( \phi \)
Bending of a Vertical Plate in Crossflow

Streamwise Velocity

\[ k = 2.0 \]

Q-Criterion (Q = 2,500)

\[ k = 2.0 \]
Siefers et al. (AIAA 2018) notes that geometrically linear deformations become invalid after $k = 0.3$.

As shown, the current method shows good agreement with established experiments and simulations.

Plate response becomes unsteady for larger values of $k$. 
Consider the setup chosen by Huang et al. (*JFM 2010*) and Hua et al. (*JFM 2014*)

- $Re = 100$
- $MR = \frac{\rho_s h}{\rho_f L} = 100$
- $\Delta x_{min} = \Delta y_{min} = 0.02L$
- Discretized with 3,200 finite elements
- FEM mesh is *pinned* at the leading edge
- $18^\circ$ crossflow to induced motion
- Thickness, $h$, is 0.01

- $S_s = \frac{EI_s}{\rho_f U^2 L^3} = 1 \times 10^3$
- $S_b = \frac{E h}{\rho_f U^2 L} = 1 \times 10^{-4}$
Waving Flag

<table>
<thead>
<tr>
<th>Reference</th>
<th>$\bar{A}$</th>
<th>St</th>
</tr>
</thead>
<tbody>
<tr>
<td>Present Work</td>
<td>0.57</td>
<td>0.22</td>
</tr>
<tr>
<td>Huang et al. <em>JFM 2010</em></td>
<td>0.58</td>
<td>0.24</td>
</tr>
<tr>
<td>Hua et al. <em>JFM 2014</em></td>
<td>0.58</td>
<td>0.24</td>
</tr>
</tbody>
</table>
Motivation/Introduction
- Mars, EDL system qualification, Simulation Capabilities

FSI Method
- Governing equations, Overview
- Immersed Boundary Method for the Compressible Navier-Stokes Equations (CFD)
- Geometrically Nonlinear Computational Structural Dynamics Solver (CSD)
- Coupling procedure

Summary of FSI Method Validation

Methods for Large-scale, Parallel CFD-CSD Coupling
- Disparate domain decomposition

Supersonic Parachute Inflation

Summary and Outlook
The method was initially developed considering a parallel CFD – serial CSD coupling framework
  - The structures consisted of a few thousand shell elements

When considering a parachute, the number of structural degrees of freedom became too large for a single processor
  - 100k and 200k+ elements in the structural domain
  - 6, 22, and 48+ million points in the fluid domain
  - 200k and 400k elements on the geometry representation (CFD and CSD)

Memory and speed optimization has been done for both solvers and the coupling framework
Parallel FSI Approach

- The CFD solver uses an octree data structure to organize the volume data
  - The geometry representation is partitioned accordingly
Parallel FSI Approach

- The CSD solver is partitioned on an unstructured mesh by ParMETIS
The problem now is partitioned in both solvers
  - But the solvers must exchange information through the geometry representation
To avoid large amounts of communication throughout the simulation, do a large amount of communication during initialization.
During initialization, pipelines between the CSD and CFD solvers are created

These are stored for the duration of the simulation
- Subsequently, information is simply passed through existing pipelines when necessary

Able to avoid re-creating pipelines every timestep under **thin-shell assumption**
- Pipelines at timestep $nt = 1$ are the same pipelines at timestep $nt = n+1$
Outline

- **Motivation/Introduction**
  - Mars, EDL system qualification, Simulation Capabilities

- **FSI Method**
  - Governing equations, Overview
  - Immersed Boundary Method for the Compressible Navier-Stokes Equations (CFD)
  - Geometrically Nonlinear Computational Structural Dynamics Solver (CSD)
  - Coupling procedure

- **Summary of FSI Method Validation**

- **Methods for Large-scale, Parallel CFD-CSD Coupling**
  - Disparate domain decomposition

- **Supersonic Parachute Inflation**

- **Summary and Outlook**
Problem Setup

- Setup is chosen in accordance with
  - **Experiments:** Sengupta et al. (AIAA 2009)
  - **Simulations:** Karagiozis et al. (JFS 2011) and Yu et al. (AIAA 2019)

- 0.8m $D_0$ DGB Parachute design is based off Reuter et al. (AIAA 2009)
  - Sub-scale Viking parachute model *with and without* a sub-scale 70° Viking capsule

![Diagram](image)

<table>
<thead>
<tr>
<th>$L_T$</th>
<th>$L_B$</th>
<th>$L_G$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\frac{x}{d} = 10.6$</td>
<td>$0.121D_0$</td>
<td>$0.042D_0$</td>
</tr>
</tbody>
</table>
Setup is chosen in accordance with

- **Experiments**: Sengupta *et al.* (AIAA 2009)
- **Simulations**: Karagiozis *et al.* (JFS 2011) and Yu *et al.* (AIAA 2019)

- 0.8m $D_0$ DGB Parachute design is based off Reuter *et al.* (AIAA 2009)
  - Sub-scale Viking parachute model *with and without* a sub-scale 70° Viking capsule
Problem Setup

- Problem resembles spacecraft entry into the upper Martian atmosphere:

**Fluid Properties**
- \( Re = \frac{\rho_\infty u_\infty d}{\mu_\infty} = 10^5 \)
- \( \mu_\infty \) via Sutherland’s law at \( T_\infty = 294.93K \)
- \( \rho_\infty = 0.0184527 \frac{kg}{m^3} \)
- \( u_\infty = 688.89 \frac{m}{s} \)
- \( M = \frac{u_\infty}{a_\infty} = 2.0 \)

**Structural Properties**
- \( E_p = 878 \text{ MPa} \)
- \( \nu = 0.33 \)
- \( h = 6.35\times10^{-5} m \)
- \( \rho_p = 614 \frac{kg}{m^3} \)
- \( d_c = 0.99\times10^{-3} m \)
- \( E_c = 43\text{ GPa} \)
- \( \rho_c = 8.27\times10^{-4} \frac{kg}{m} \)

- Setup time: virtually none after CSD mesh is created
Problem Setup

- Center of the vent hole is at (0,0,0)
- Domain: \([-6.25D_0, 6.25D_0] \times [-6.25D_0, 6.25D_0] \times [-6.25D_0, 6.25D_0]\)
- Base case: \(\Delta x_{\text{min}} = \Delta y_{\text{min}} = D_0/164\)

- 600 geometrically nonlinear cables elements are used for the suspension lines
  - Fixed at point P

- 108,000 geometrically nonlinear shell elements resolve the disk and canopy
Problem Setup

- Center of the vent hole is at (0,0,0)
- Domain: \([-6.25D_0, 6.25D_0] \times [-6.25D_0, 6.25D_0] \times [-6.25D_0, 6.25D_0]\)
- Base case: \(\Delta x_{min} = \Delta y_{min} = D_0/164\)

- 600 geometrically nonlinear cables elements are used for the suspension lines
  - Fixed at point P
- Simulation initial condition (unstressed)

- 108,000 geometrically nonlinear shell elements resolve the disk and canopy
Problem Setup

- Structural mesh based off simulations by Derkevorkian et al. *(AIAA 2019)*
  - Elements along seams are thickened by a factor of 4 to represent the stitching pattern used in manufacturing of the canopy
  - Finely resolving these regions also helps capture the stress discontinuities across the seams
Case 1: Uniform Flow (no capsule)

Streamwise velocity

Temperature field
Case 1: Uniform Flow (no capsule)

Simulation of sub-scale DGB Parachute

Full-scale ASPIRE Test flight

*Not a direct comparison

(NASA/JPL)
Case 1: Uniform Flow (no capsule)

- The cables are not resolved in the CFD volume mesh
  - Nor do they experience any external loading
  - This may lead to large period, large amplitude swaying of the cables

- The cables, as well as the canopy, start the simulation in an unstressed/un-tensioned state

- How much do the cable dynamics affect simulations of inflation versus reality?
Case 2: Leading Viking-type Capsule

Streamwise velocity
Outline

- Motivation/Introduction
  - Mars, EDL system qualification, Simulation Capabilities

- FSI Method
  - Governing equations, Overview
  - Immersed Boundary Method for the Compressible Navier-Stokes Equations (CFD)
  - Geometrically Nonlinear Computational Structural Dynamics Solver (CSD)
  - Coupling procedure

- Summary of FSI Method Validation

- Methods for Large-scale, Parallel CFD-CSD Coupling
  - Disparate domain decomposition

- Supersonic Parachute Inflation

- Summary and Outlook
Summary:

- An overview of a validated numerical FSI method was presented
  - Cartesian IBM CFD
  - Nonlinear CSD

- The details of the weak, parallel coupling algorithm between the CFD and CSD solvers was outlined

- FSI validation cases were presented

- The FSI method was finally applied to the simulation of parachute inflation in the upper Martian atmosphere
Summary and Outlook

- **Outlook:**
  - Apply porous material boundary conditions on the canopy
  - Implement efficient contact algorithms for robustness and more interesting starting conditions
  - Utilize adaptive mesh refinement (AMR) in the simulations
  - Start to compare against ASPIRE missions
Questions?