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

Jonathan Boustani[*,+], Michael F. Barad[+], Cetin C. Kiris[+], Christoph Brehm[*]

Program Director, Dr. Suzanne Smith
Grant Number, RIDG-17-005

HPC Resources Provided by

[*] *Department of Mechanical Engineering, University of Kentucky, Lexington, KY, 40506, USA*

[+] *Computational Aerosciences Branch, NASA Ames, Moffet Field, CA, 94035, USA*

Outline

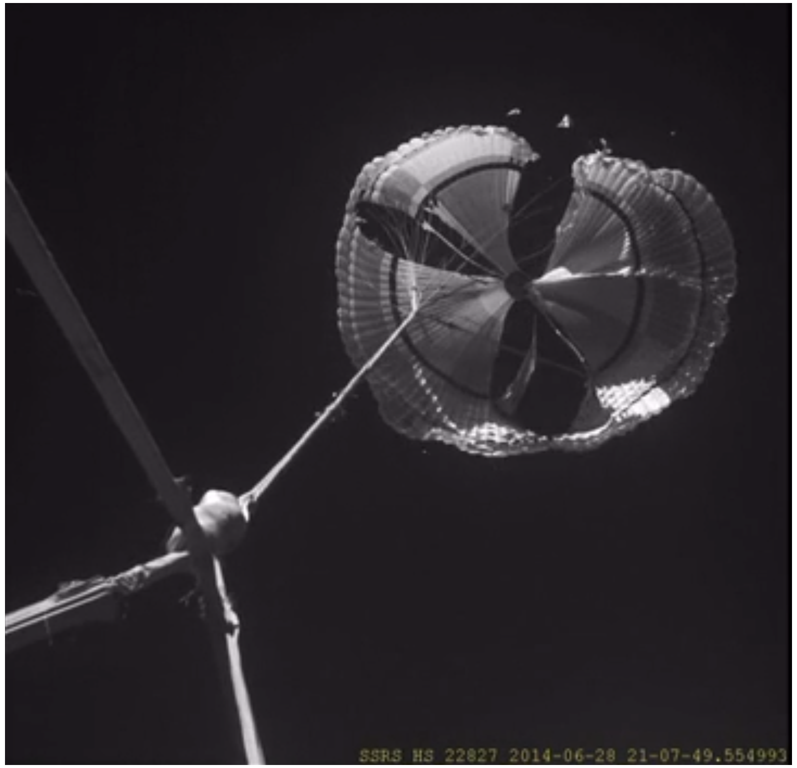
- ❑ **Motivation/Introduction**
  - Mars, EDL system qualification, Simulation Capabilities
- ❑ **FSI Method**
  - Governing equations
  - Immersed Boundary Method for the Compressible Navier-Stokes Equations (CFD)
  - Geometrically Nonlinear Computational Structural Dynamics Solver (CSD)
  - Coupling procedure
- ❑ **Extended Validation for Fluid-Structure Interaction Problems**
- ❑ **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
  - Immersed Boundary Method for the Compressible Navier-Stokes Equations (CFD)
  - Geometrically Nonlinear Computational Structural Dynamics Solver (CSD)
  - Coupling procedure
- **Extended Validation for Fluid-Structure Interaction Problems**
- **Methods for Large-scale, Parallel CFD-CSD Coupling**
  - Disparate domain decomposition
- **Supersonic Parachute Inflation**
- **Summary and Outlook**

Motivation

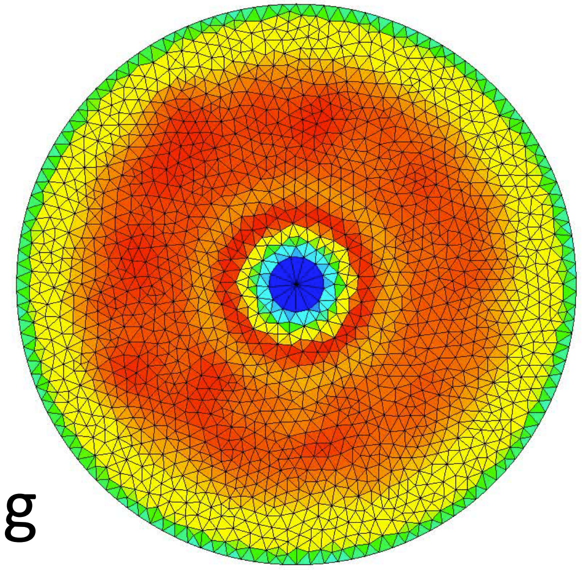
- MSL EDL system was requalified
  - Payload weight, canopy size, and landing altitude exceeded those established by heritage Viking mission
    (Sengupta *et al*. AIAA 2007,2009, Way *et al*. IEEE 2006)
- NASA's mission to Mars will eventually require EDL re-qualification
  - For hardware and humans required for sustained settlements, more demanding landing objectives
- LDSD project
  - Supersonic ringsail parachute
  - Low-Density Supersonic Decelerator



(NASA)

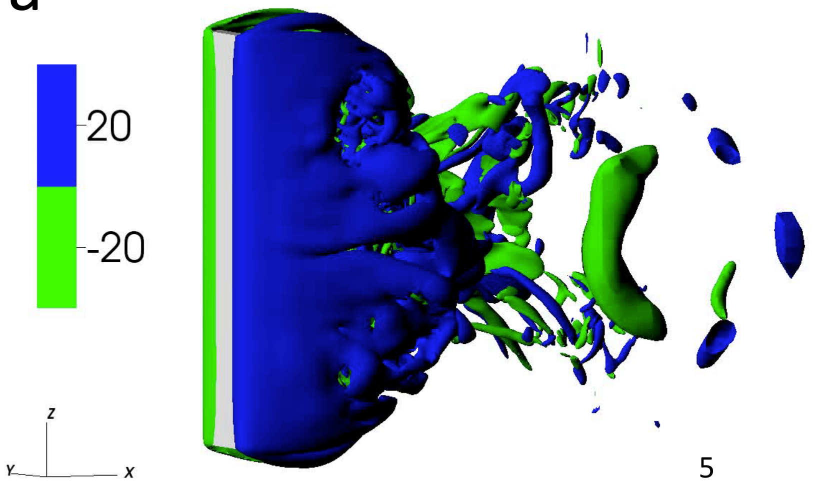
Introduction

- Previously introduced and validated a method for simulating the large, geometrically nonlinear deformations of very thin shell structures (Boustani *et al.* SciTech 2019)



- This work is an extension of these capabilities to solving large-scale FSI problems in high-speed flows within a parallel computing environment

- End goal is to simulate supersonic parachute deployment



Outline

- **Motivation/Introduction**
  - Mars, EDL system qualification, Simulation Capabilities
- **FSI Method**
  - Governing equations
  - Immersed Boundary Method for the Compressible Navier-Stokes Equations (CFD)
  - Geometrically Nonlinear Computational Structural Dynamics Solver (CSD)
  - Coupling procedure
- **Extended Validation for Fluid-Structure Interaction Problems**
- **Methods for Large-scale, Parallel CFD-CSD Coupling**
  - Disparate domain decomposition
- **Supersonic Parachute Inflation**
- **Summary and Outlook**

Governing Equations

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

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

$$\mathbf{W} = \left[\ \rho, \rho u, \rho v, \rho w, \rho e_t\ \right]^T$$

- The structural regime considers the Total Lagrangian equations of motion

$$\int_{^0V} {}_0\boldsymbol{S}_{ij}\delta_0\boldsymbol{\epsilon}_{ij} d^0V + \int_{^0V} {}^t_0\boldsymbol{S}_{ij}\delta_0\boldsymbol{\eta}_{ij} d^0V = {}^{t+\Delta t}\boldsymbol{\mathcal{R}} - \int_{^0V} {}^t_0\boldsymbol{S}_{ij}\delta_0\boldsymbol{e}_{ij} d^0V$$

- Partitioned solution involves solving strong and weak solutions together

Coupling Conditions

- The coupling conditions between the two regimes enforce the continuity of loads across the shared boundary

$$\mathbf{t}_{structure}(\overline{\mathbf{x}}_b(t), t) = \mathbf{t}_{fluid}(\overline{\mathbf{x}}_b(t), t)$$

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

$$\overline{\mathbf{x}}_b(t) = \mathbf{x}_{fluid}(t) = \mathbf{x}_{structure}(t), \text{ and}$$
$$\dot{\overline{\mathbf{x}}}_b(t) = \dot{\mathbf{x}}_{fluid}(t) = \dot{\mathbf{x}}_{structure}(t) \;\; \forall t \geq 0$$

FSI Method

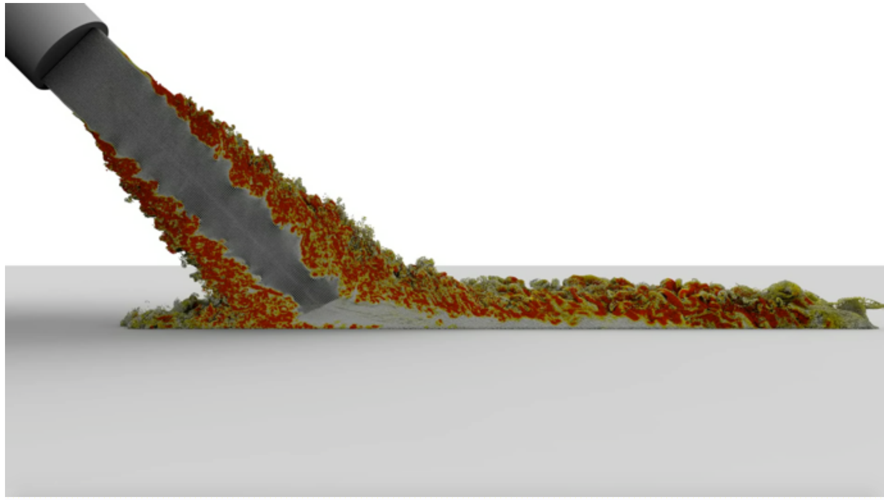
- The method used in this work couples together

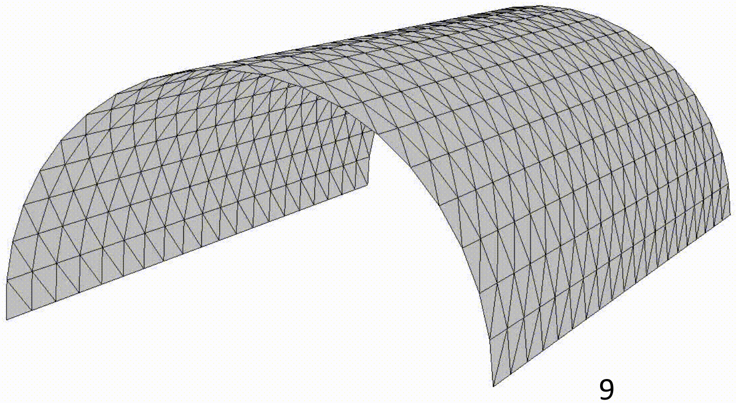
  I. A structured Cartesian, higher-order, sharp immersed boundary method for the compressible Navier-Stokes equations

     *Brehm, C., Fasel, H., **JCP 2013***
     *Brehm, C., Hader, C., Fasel, H., **JCP 2015***
     *Brehm, C., Barad, M. F., Kiris, C. C., **JCP 2018***

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

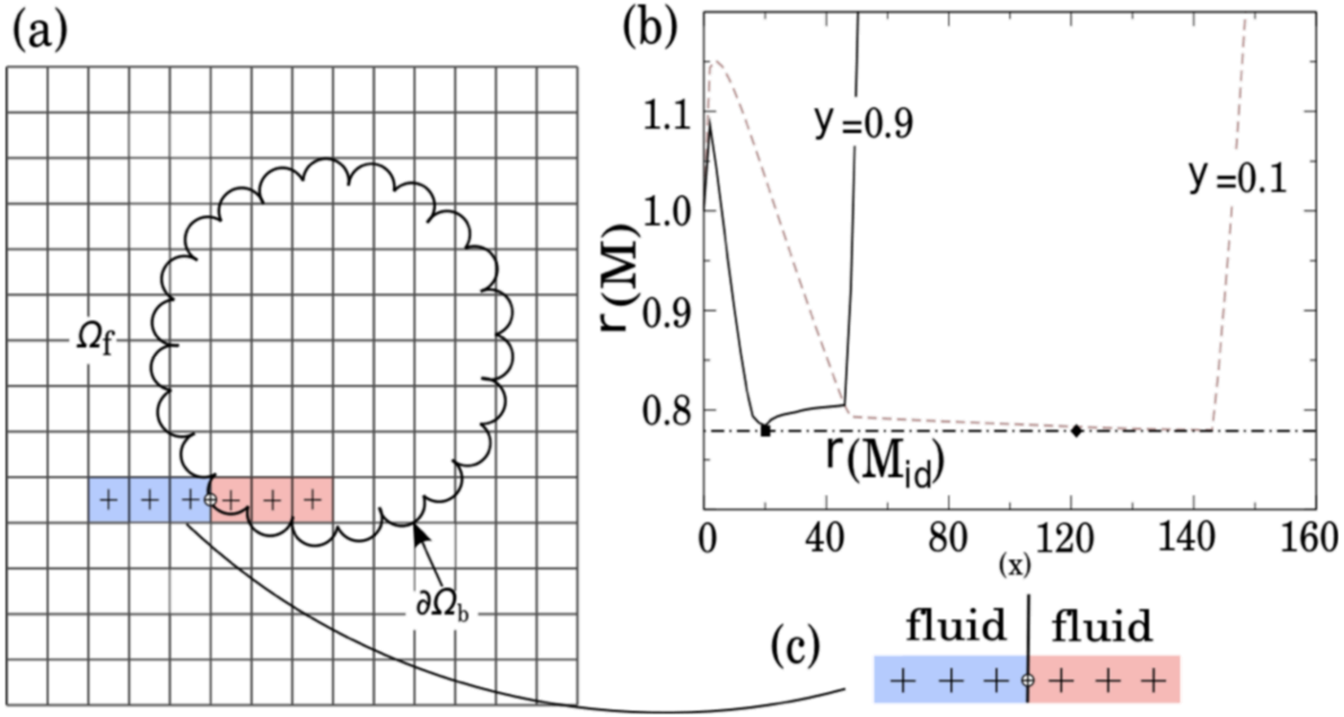
     *Boustani et al., **AIAA SciTech 2019***





Higher-Order IBM

- FD stencils are locally optimized considering the local flow conditions and boundary distance
  - Improved stability
- Compressible Navier-Stokes are solved with the 4^{th}- order time explicit Runge-Kutta scheme
- WENO5 is used for the convective terms to deal with flow discontinuities

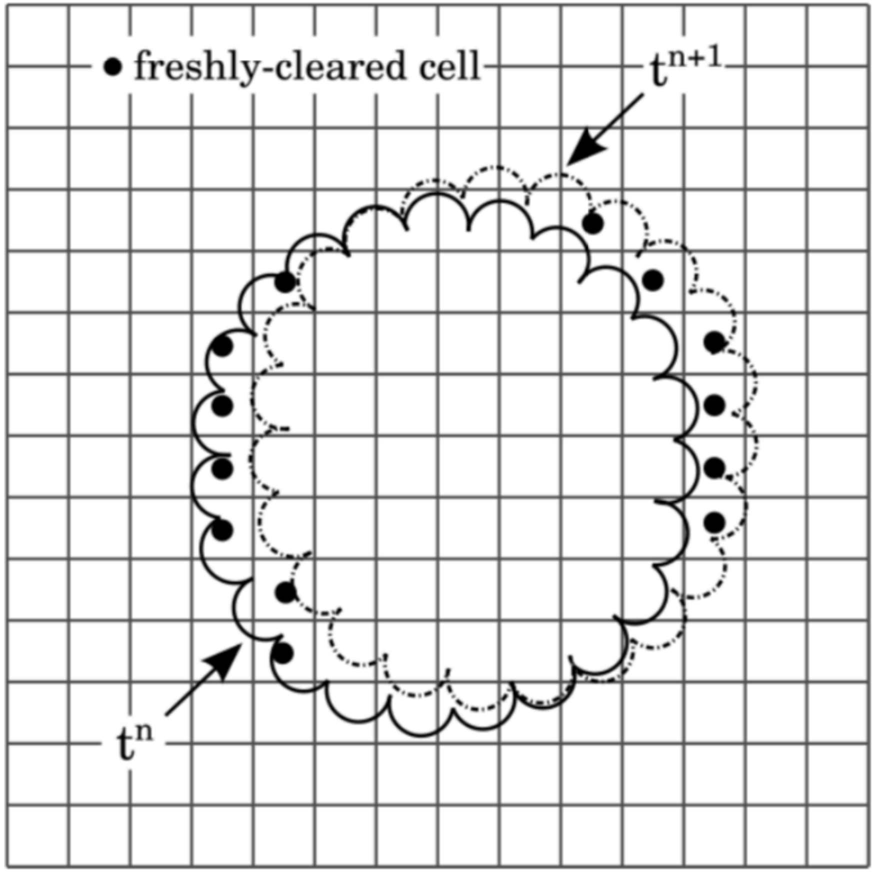


Higher-Order IBM

- ‘Sharp’ classification comes from boundary conditions being enforced directly at grid-line intersection points

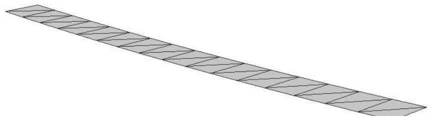
- Advantageous for thin geometries
  - No valid data is needed inside the geometry
  - Now need to deal with freshly-cleared cells (FCCs)

- FCCs have no valid-time history
  - Must interpolate from the surrounding flow
  - Use canonical ENO selection in high-speed flows



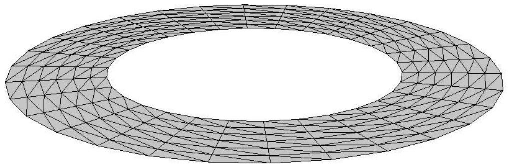
Structural Solver

- Element formulations used:
  - Geometrically nonlinear MITC3 triangular shell element
  - Geometrically nonlinear generic cable element



- In this work, the St. Venant-Kirchoff hyperelastic strain-energy function is used

- Time integration is performed with the implicit Newmark-β scheme
  - Nonlinear solution is obtained via Newton-Raphson iteration

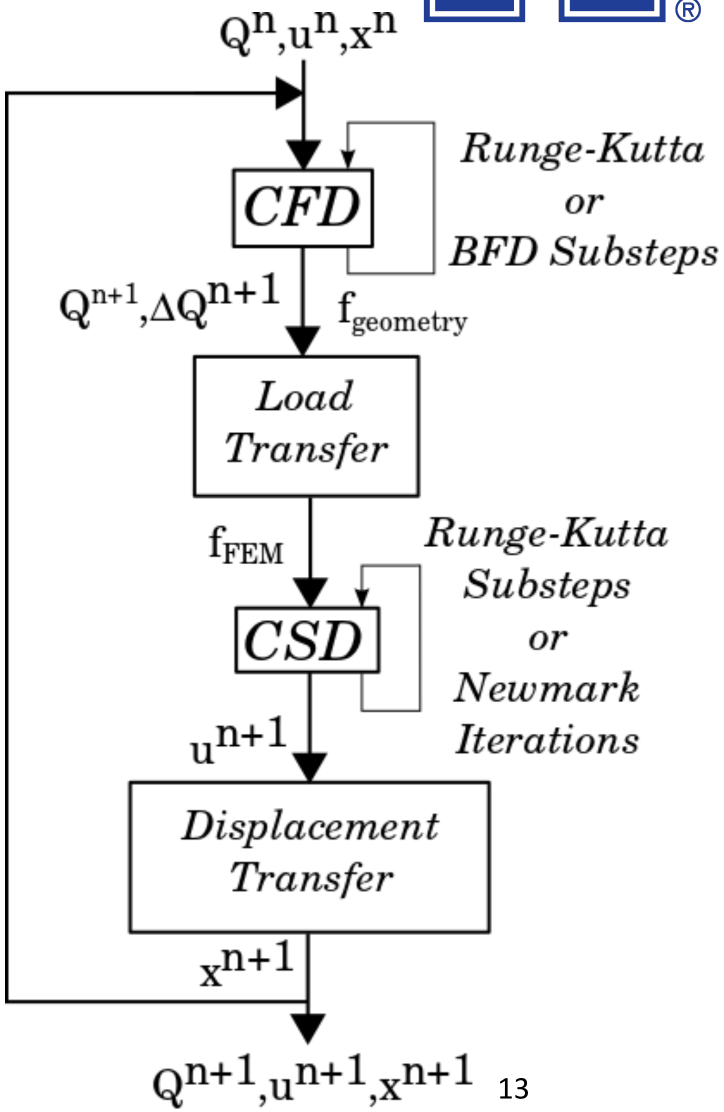


CFD-CSD Coupling

- The CFD and CSD solvers are weakly coupled
  - The solution procedure is **partitioned**

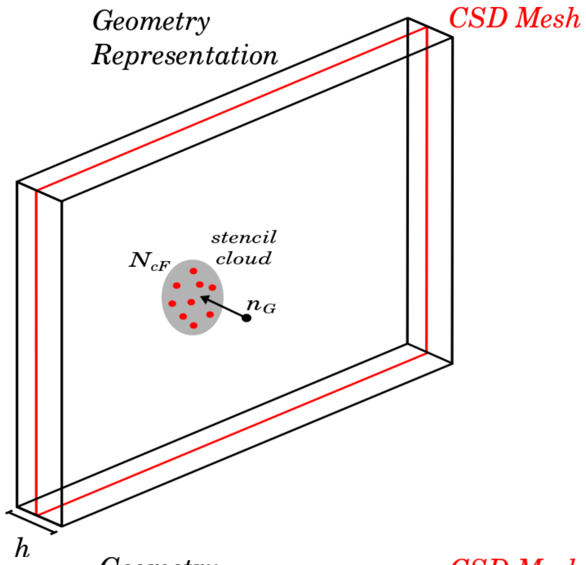
- An auxiliary and mass-less, or phantom, representation of the geometry with a finite thickness is used in the CFD solver

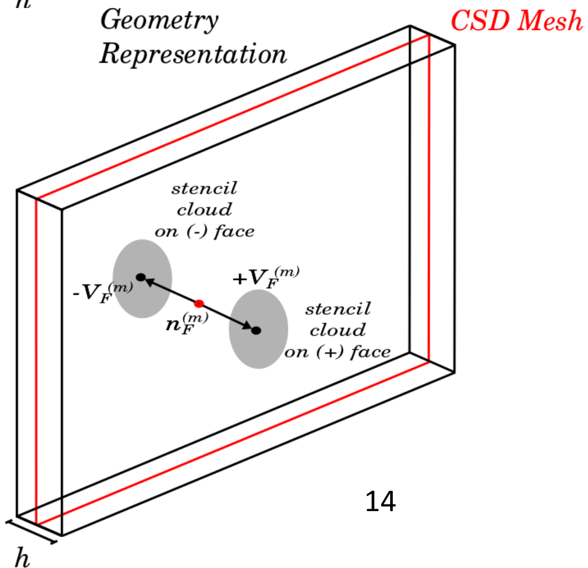
- The coupling conditions are enforced at the artificial interface between the geometry representation and the infinitesimal thickness CSD mesh



CFD-CSD Coupling

- The CFD and CSD solvers are weakly coupled
  - The solution procedure is **partitioned**
- 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 at the artificial interface between the geometry representation and the infinitesimal thickness CSD mesh





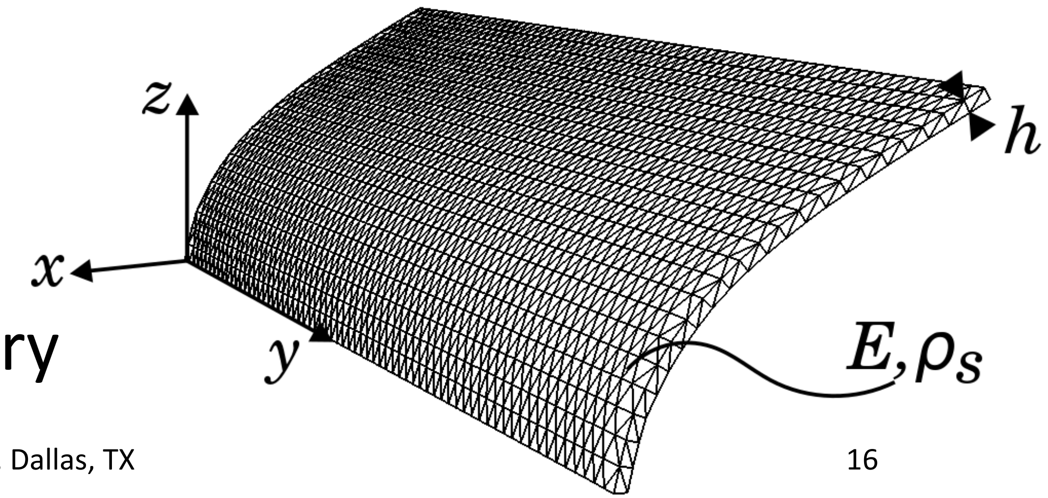
Outline

- ❑ **Motivation/Introduction**
  - Mars, EDL system qualification, Simulation Capabilities
- ❑ **FSI Method**
  - Governing equations
  - Immersed Boundary Method for the Compressible Navier-Stokes Equations (CFD)
  - Geometrically Nonlinear Computational Structural Dynamics Solver (CSD)
  - Coupling procedure
- ❑ **Extended Validation for Fluid-Structure Interaction Problems**
- ❑ **Methods for Large-scale, Parallel CFD-CSD Coupling**
  - Disparate domain decomposition
- ❑ **Supersonic Parachute Inflation**
- ❑ **Summary and Outlook**

Bending of a Vertical Plate in Crossflow

- Vertical plate of height H is *clamped* along bottom edge
- All parameters chosen in accordance with
  - **Simulations**: Hu and Wang (JAFM 2016), and Seidel *et al.* (AIAA 2018)
  - **Experiments**: Womack and Seidel (AIAA 2014), and Siefers *et al.* (AIAA 2018)

- Exposed to viscous crossflow $\mathbf{U} = (U, 0,0)^T$
- Domain: [-20H,25H] × [-9H,9H] × [0,9H]
- $\Delta x_{min} = \Delta y_{min} = \mathrm{H}/25$
- No-slip wall on plate, slip wall on x-y boundary



Bending of a Vertical Plate in Crossflow

- For comparison with Hu and Wang (JAFM 2016) and Womack and Seidel (AIAA 2014), Siefers *et al.* (AIAA 2018) introduced the

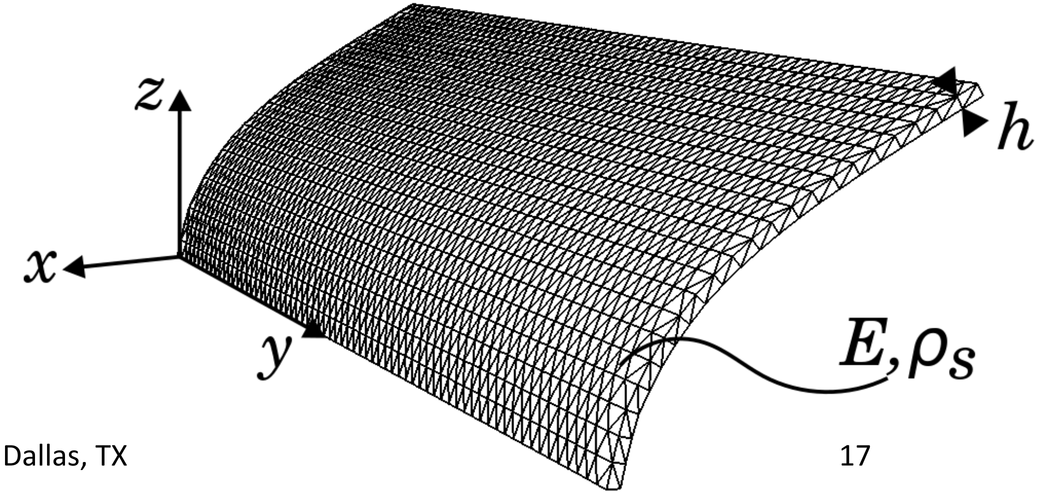
  1. Mean chord angle

$$\emptyset = \tan^{-1}\left(\frac{\delta_x}{H-\delta_z}\right)$$

  2. Normalized curvature

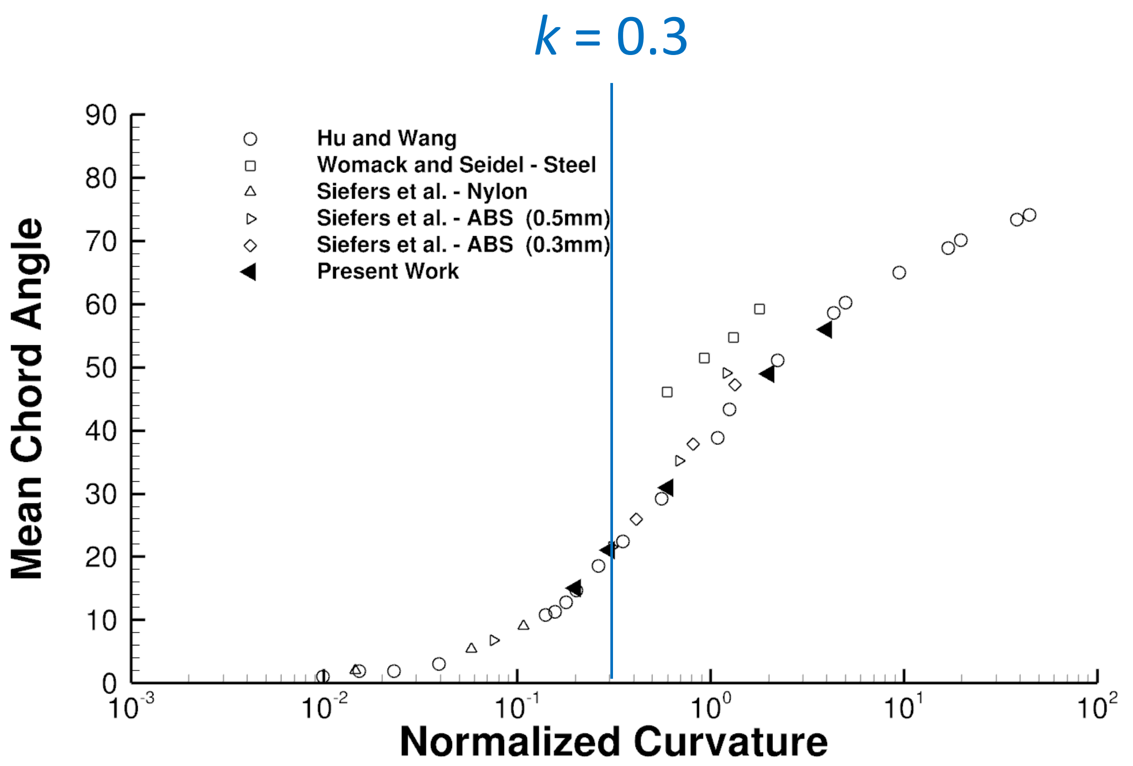
$$k = \frac{qH^3}{Eh^3}$$

- These parameters reduce the solution to a single variable, $\emptyset$



Bending of a Vertical Plate in Crossflow

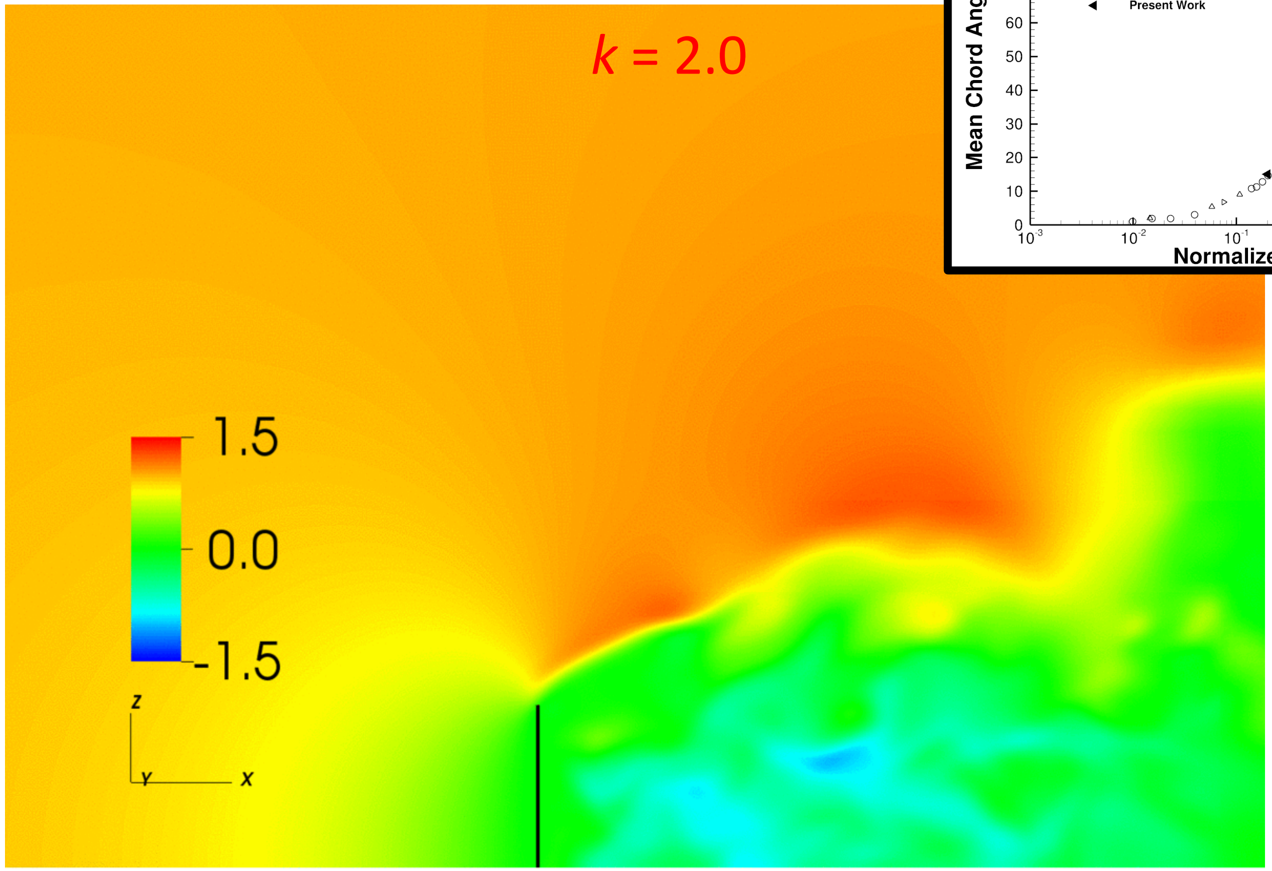
- Siefers *et al.* (AIAA 2018) notes that geometrically linear deformations become invalid after $k = 0.3$
- As shown, the current method shows good agreement with established experiments and simulations
- Plate response becomes unsteady for larger values of k

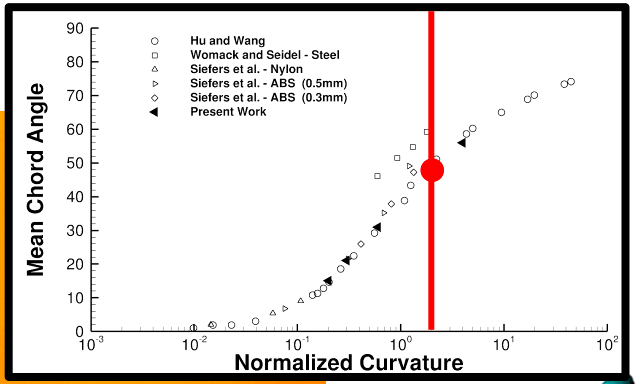


Bending of a Vertical Plate in Crossflow

Streamwise Velocity

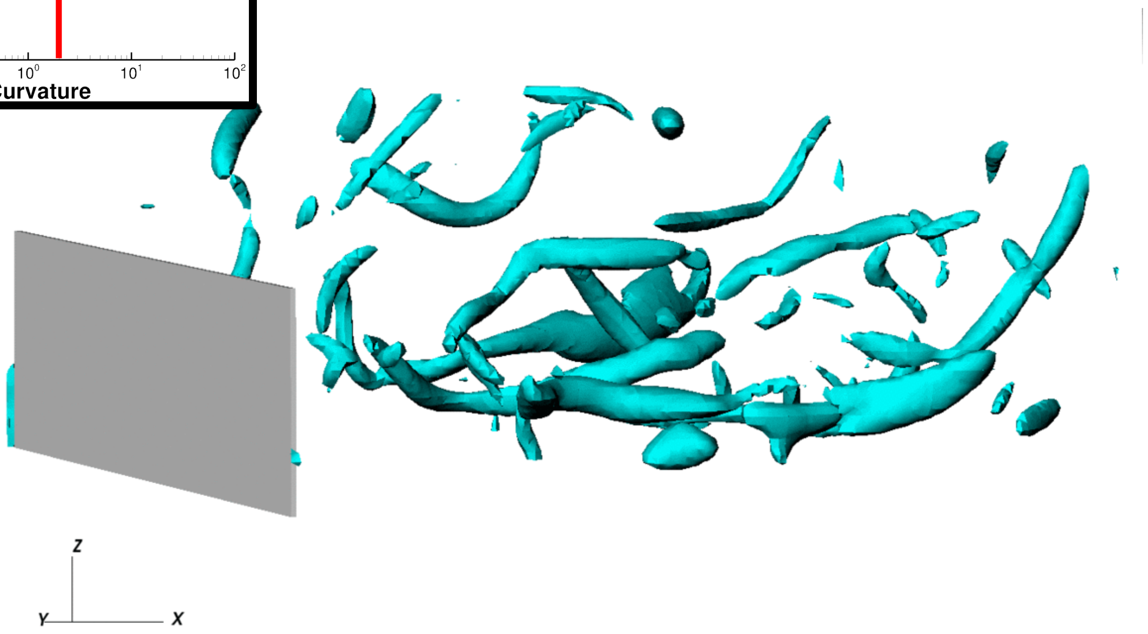
k = 2.0





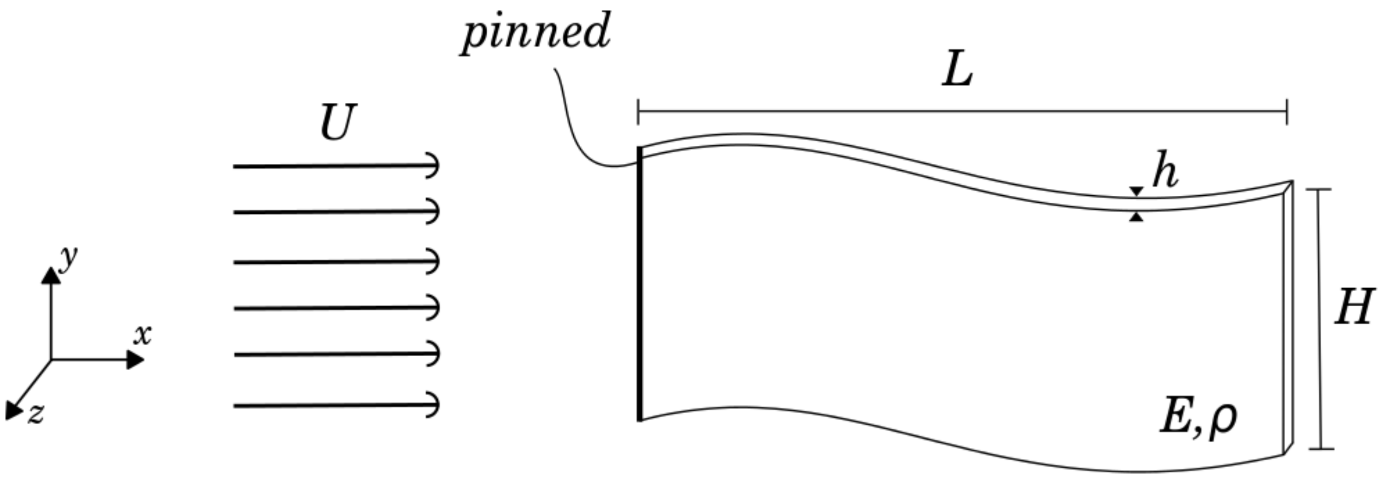
Q-Criterion (Q = 2,500)

k = 2.0



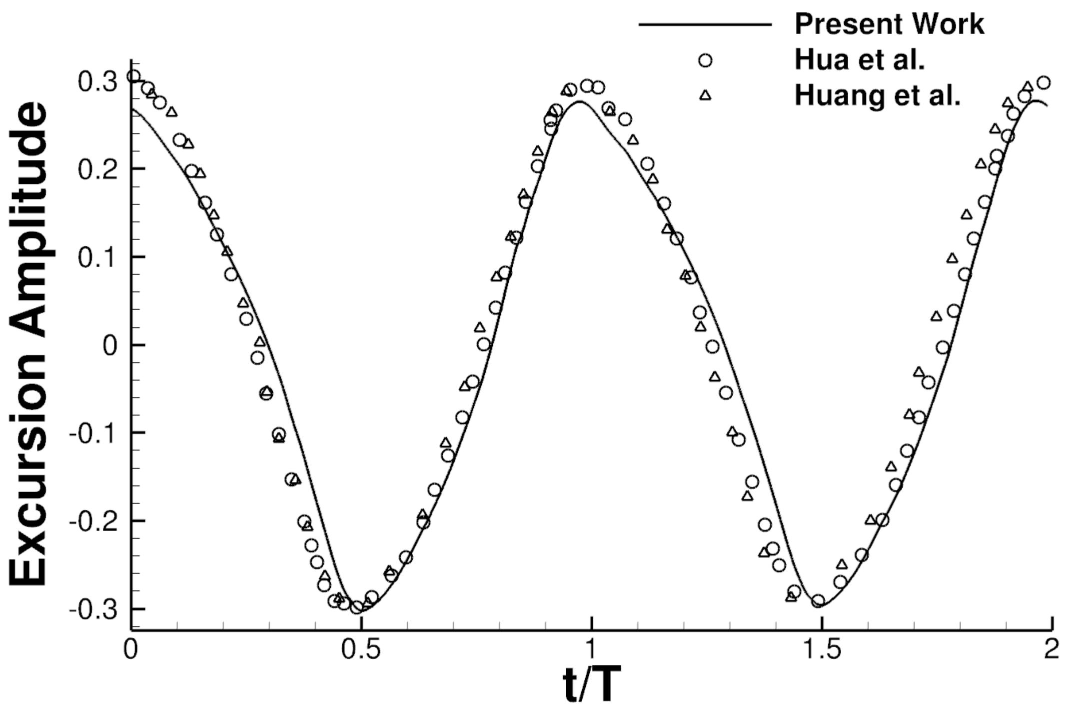
Waving Flag

- 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° crossflow to induced motion
  - Thickness, *h*, is 0.01
  - $\boldsymbol{S_s = \frac{EI_s}{\rho_f U^2 L^3} = 1{\times}10^3}$,
  - $\boldsymbol{S_b = \frac{Eh}{\rho_f U^2 L} = 1{\times}10^{-4}}$



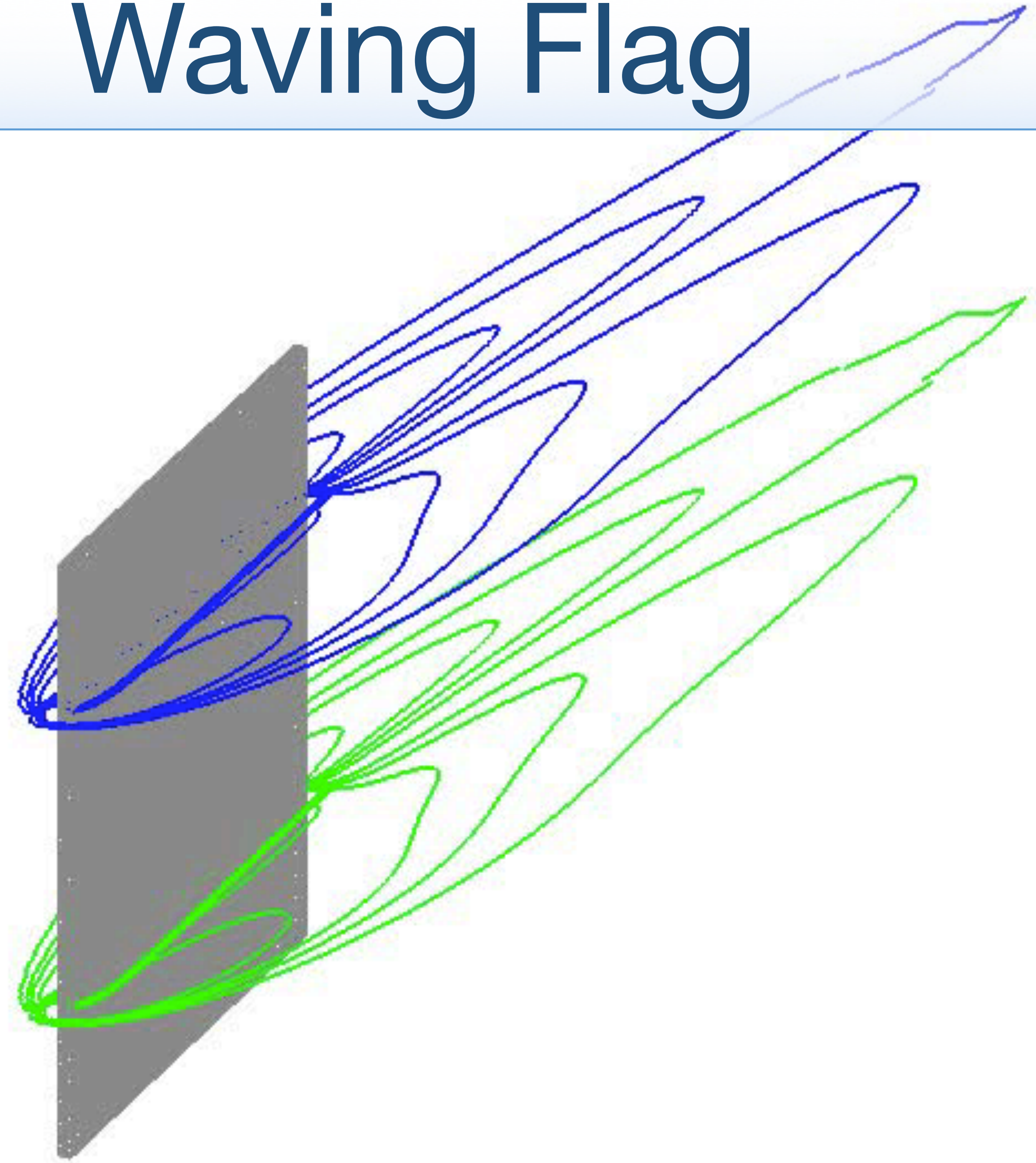
Waving Flag

Reference	$\bar{A}$	St
Present Work	**0.57**	**0.22**
Huang *et al.* *(JFM 2010)*	0.58	0.24
Hua *et al.* *(JFM 2014)*	0.58	0.24



- As shown, good agreement is obtained both in terms of the excursion amplitude $\frac{(\delta_{z,max}-\delta_{z,min})}{L}$ and the Strouhal number $f\,\frac{U}{L}$

Waving Flag



Outline

- ❑ **Motivation/Introduction**
  - Mars, EDL system qualification, Simulation Capabilities
- ❑ **FSI Method**
  - Governing equations
  - Immersed Boundary Method for the Compressible Navier-Stokes Equations (CFD)
  - Geometrically Nonlinear Computational Structural Dynamics Solver (CSD)
  - Coupling procedure
- ❑ **Extended Validation for Fluid-Structure Interaction Problems**
- ❑ **Methods for Large-scale, Parallel CFD-CSD Coupling**
  - Disparate domain decomposition
- ❑ **Supersonic Parachute Inflation**
- ❑ **Summary and Outlook**

Large FSI Problems

- In Boustani *et al.* SciTech 2019, the structures consisted of a few thousand shell elements
  - This allowed a *parallel CFD – serial CSD* coupling

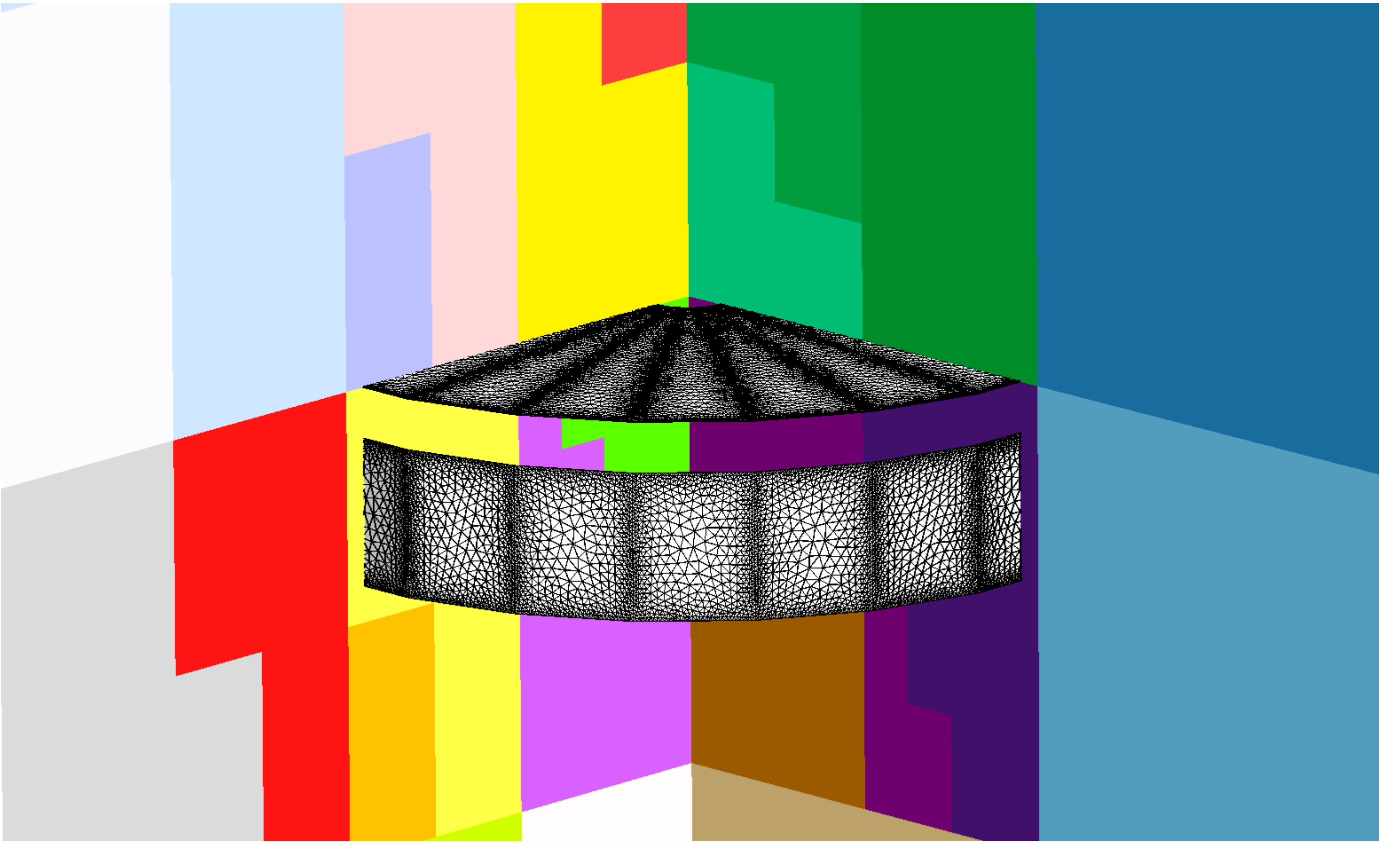
- When considering a parachute geometry, the number of degrees of freedom requires parallel computing
  - The *parallel CFD – parallel CSD* coupling requires a complex communication pattern
  - When dealing with large-scale problems, minimize memory and overhead

- What happens when the CFD and CSD partitions are disparate?
  - Expected in weakly coupled FSI algorithms

Parallel FSI Algorithm

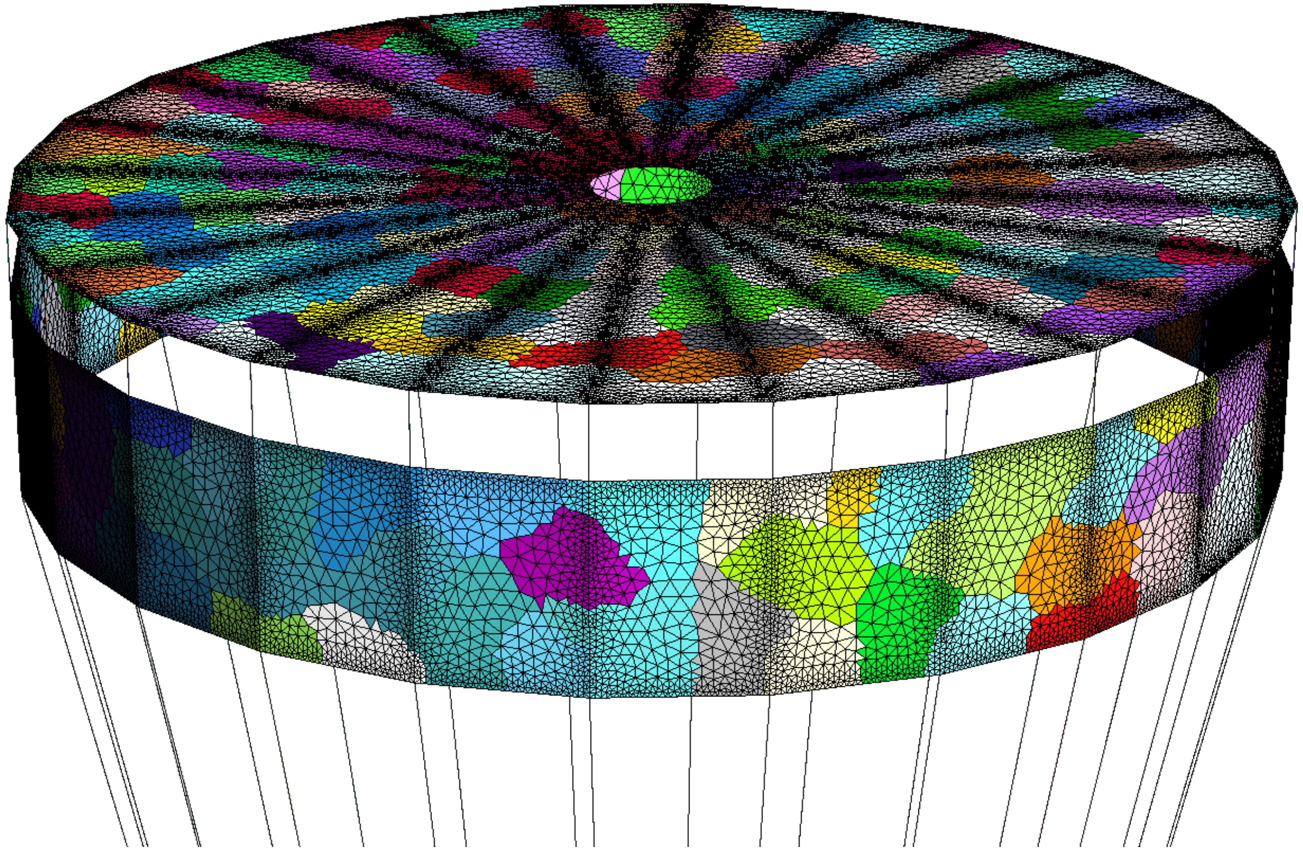
I. The CFD solver uses an octree data structure to organize the volume data

- **The geometry representation is partitioned accordingly**



Parallel FSI Algorithm

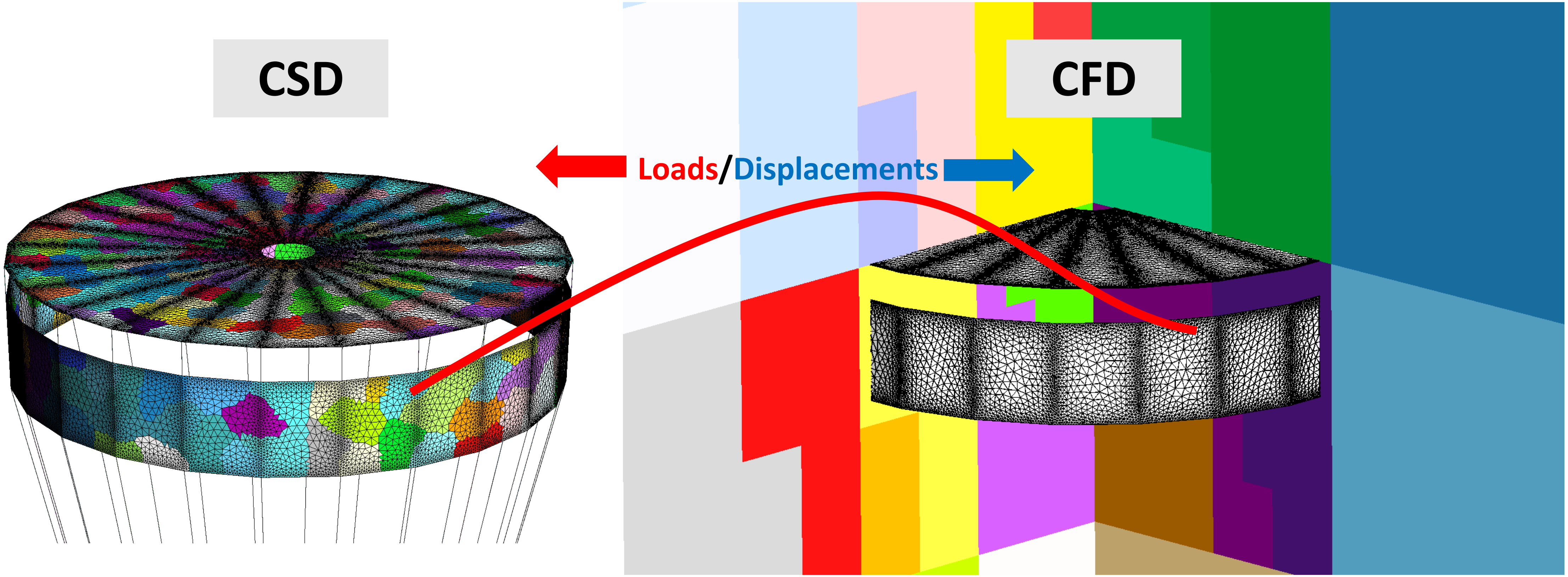
II. The CSD solver is partitioned on an unstructured mesh by ParMETIS



Parallel FSI Algorithm

Aside:

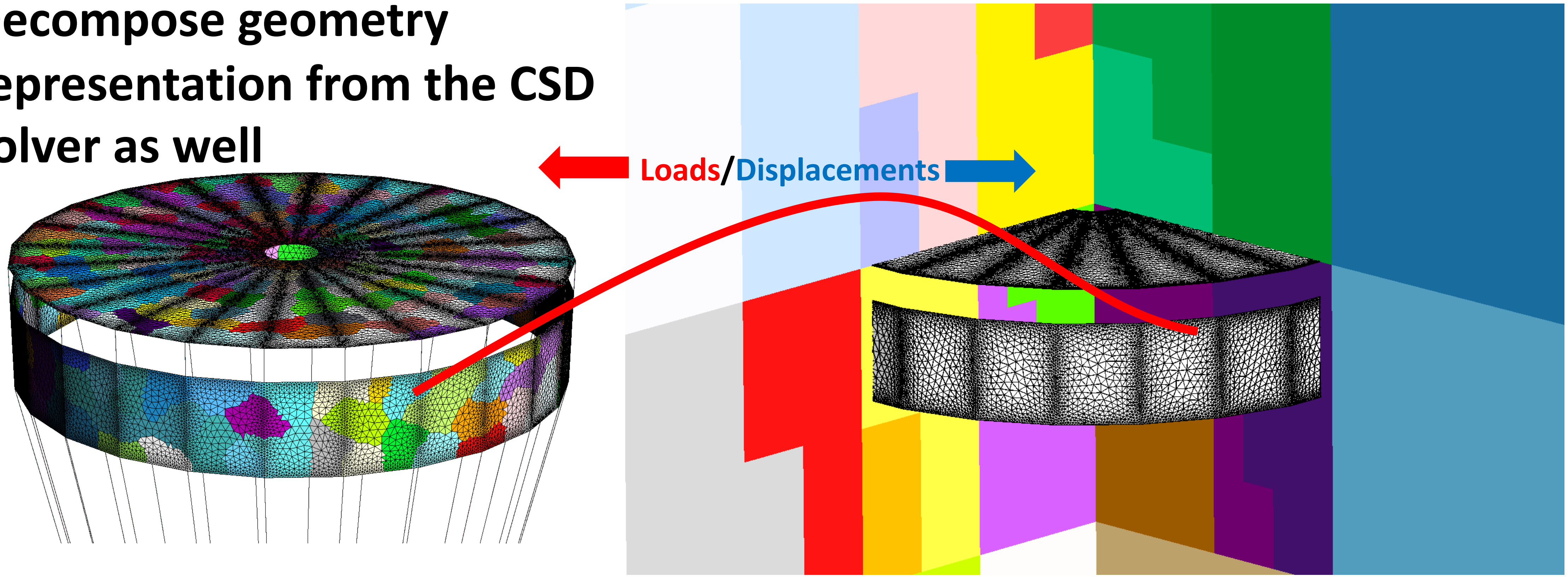
- ❑ How does process ‘*m*’ get/transfer loads/displacements to/from process ‘*n*’?



Parallel FSI Algorithm

Aside:

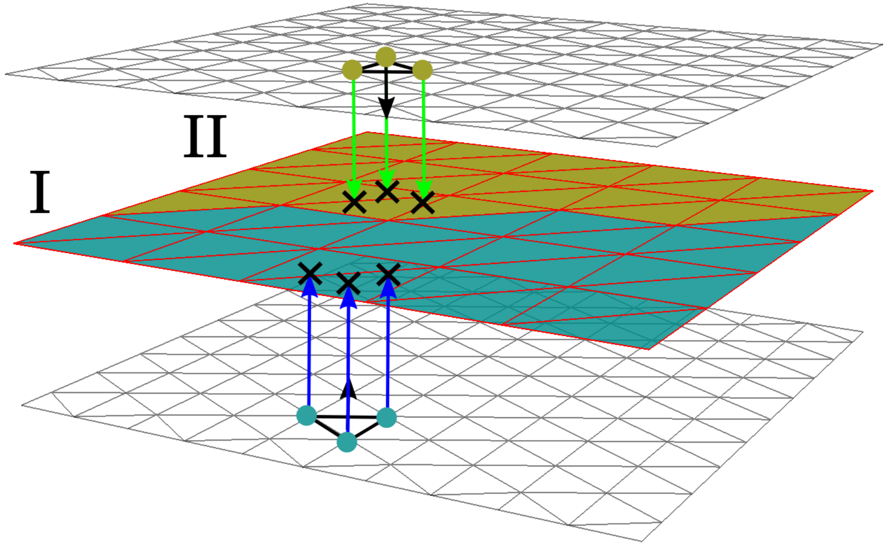
- How does process '*m*' get/transfer loads/displacements to/from process '*n*'?
- **Decompose geometry representation from the CSD solver as well**



Parallel FSI Algorithm

Aside:

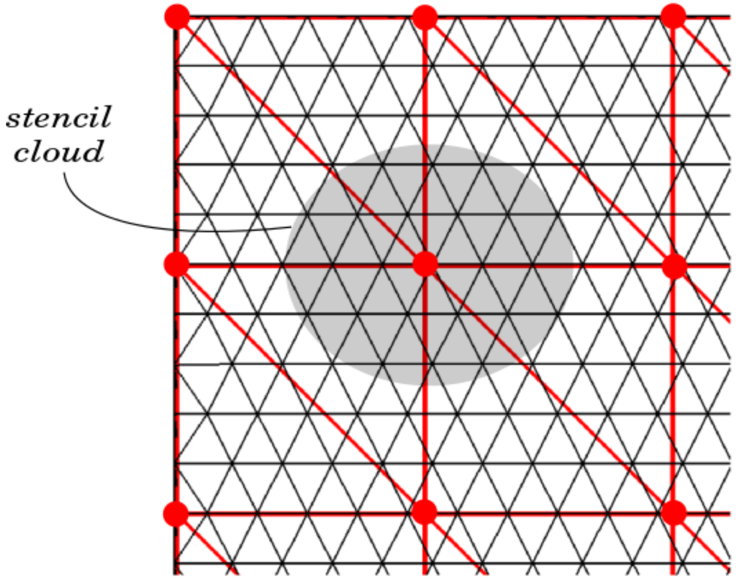
- ❑ Ray-triangle intersection is used to identify elements in the geometry representation laying directly ‘above/below’ a CSD partition
  - ○ Ray intersect a CSD element belonging to a partition and are stored **uniquely** by that partition



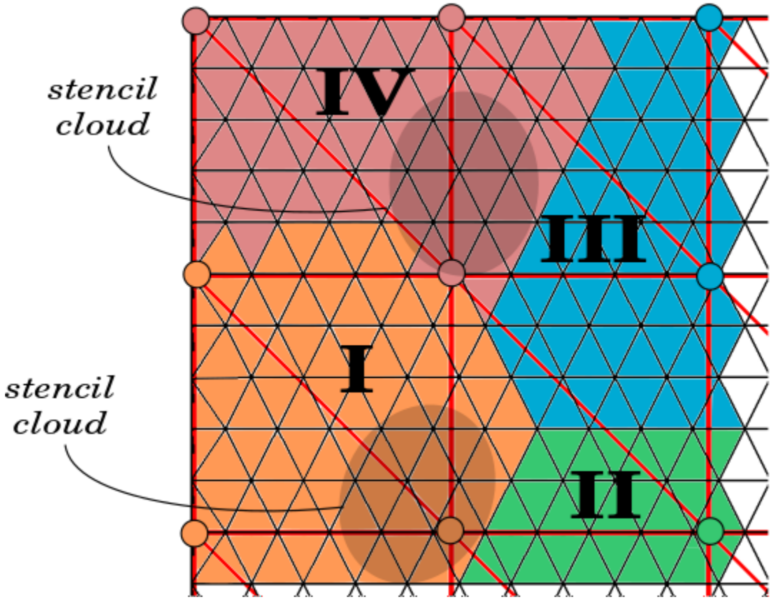
Parallel FSI Algorithm

III. Load and displacement transfer stencils are computed between the geometry representation and CSD mesh within the defined partitions

- Stencils are limited a single partition



Serial CSD Displacement Stencil



Parallel CSD Displacement Stencil

Parallel FSI Algorithm

- ❑ Using this algorithm, each process only stores its portion(s) of the CFD volume mesh, geometry representation, and the CSD mesh
  - Need to communicate to other processors is reduced greatly
  - Memory requirements are less demanding

- ❑ It is clear that the geometry representation is stored twice
  - Once when partitioned by the CFD solver via volume decomposition
  - Once when partitioned by the CSD solver via ray-triangle intersection
  - No guarantee that these partitions are the same

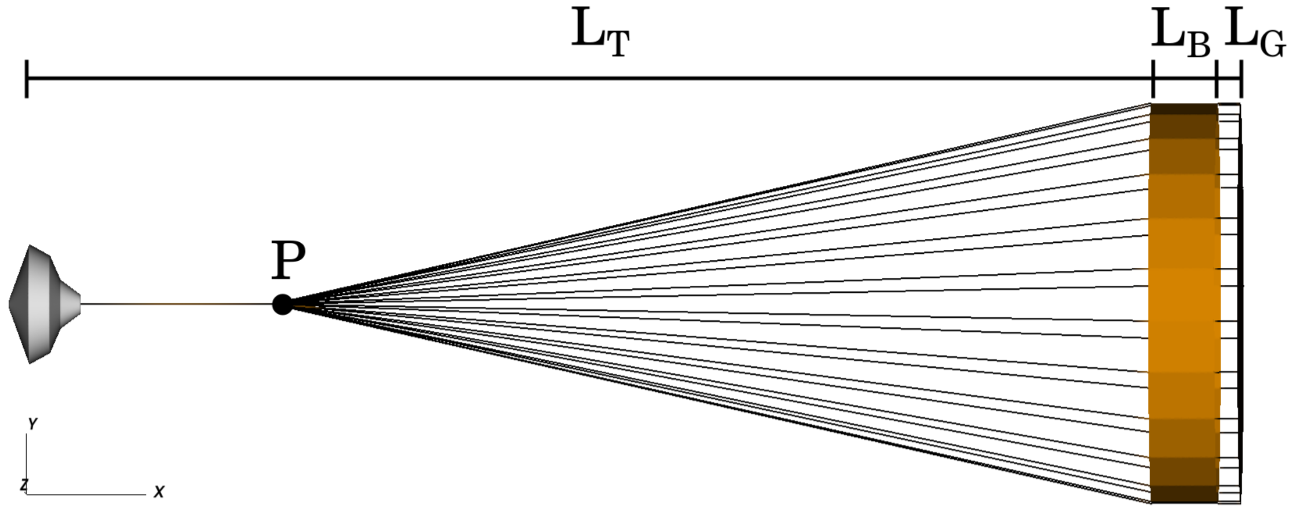
- ❑ Best case scenario is a shared, infinitesimal thickness representation of the CSD mesh and geometry representation

Outline

- **Motivation/Introduction**
  - Mars, EDL system qualification, Simulation Capabilities
- **FSI Method**
  - Governing equations
  - Immersed Boundary Method for the Compressible Navier-Stokes Equations (CFD)
  - Geometrically Nonlinear Computational Structural Dynamics Solver (CSD)
  - Coupling procedure
- **Extended Validation for Fluid-Structure Interaction Problems**
- **Methods for Large-scale, Parallel CFD-CSD Coupling**
  - Disparate domain decomposition
- **Supersonic Parachute Inflation**
- **Summary and Outlook**

Problem Setup

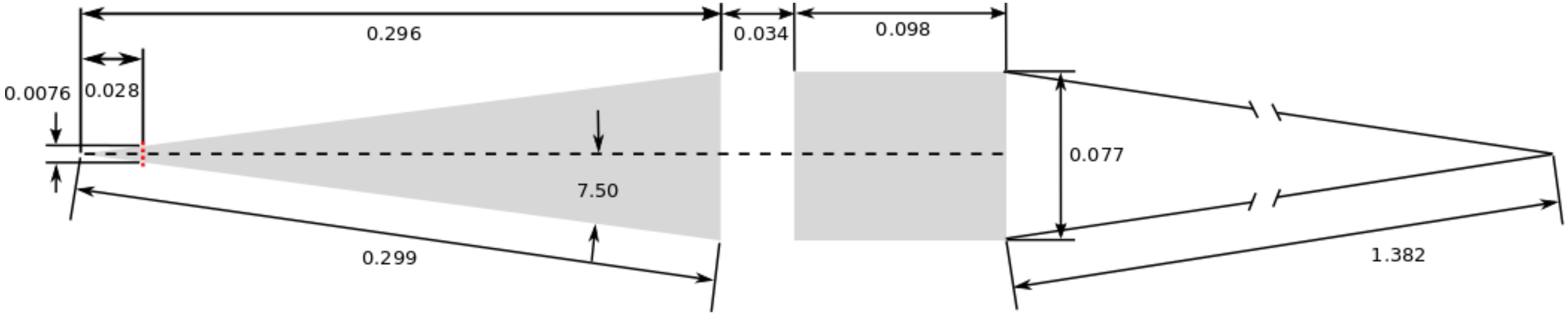
- Setup is chosen in accordance with
  - **Experiments:** Sengupta *et al.* (AIAA 2009)
  - **Simulations:** Karagiozis *et al*. (JFS 2011) and Yu *et al.* (AIAA 2019)
- 0.8m D_0 DGB Parachute design is based off Reuter *et al.* (AIAA 2009)
  - Sub-scale Viking parachute model ***with and without*** a sub-scale 70° Viking capsule



L_T	L_B	L_G
$\frac{x}{d} = 10.6$	$0.121D_0$	$0.042D_0$

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



Problem Setup

- Problem resembles spacecraft entry into the upper Martian atmosphere:

Fluid Properties

- $Re = \frac{\rho_\infty u_\infty d}{\mu_\infty} = 10^5$
- μ_∞ via Sutherland's law at $T_\infty = 294.93K$
- $\rho_\infty = 0.0184527 \frac{kg}{m^3}$
- $u_\infty = 688.89 \frac{m}{s}$
- $M = \frac{u_\infty}{a_\infty} = 2.0$

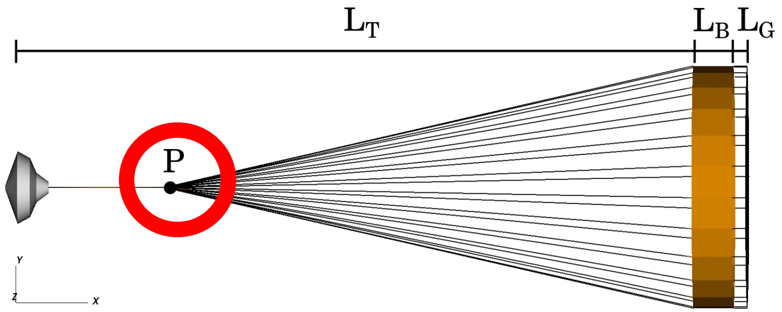
Structural Properties

- $E_p = 878\ MPa$
- $\nu = 0.33$
- $h = 6.35{\times}10^{-5} m$
- $\rho_p = 614 \frac{kg}{m^3}$
- $d_c = 0.99{\times}10^{-3} m$
- $E_c = 43 GPa$
- $\rho_c = 8.27{\times}10^{-4} \frac{kg}{m}$

Problem Setup

- Center of the vent hole is at (0,0,0)
- Domain: [-6.25D_0, 6.25D_0] × [-6.25D_0, 6.25D_0] × [-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

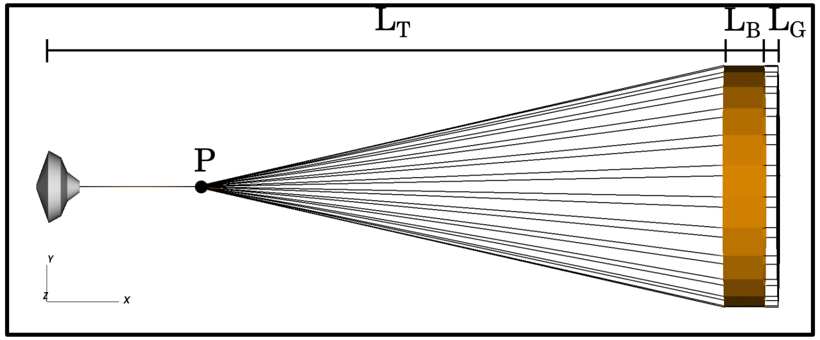


- 108,000 geometrically nonlinear shell elements resolve the disk and canopy

Problem Setup

- Center of the vent hole is at (0,0,0)
- Domain: [-6.25D_0, 6.25D_0] × [-6.25D_0, 6.25D_0] × [-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

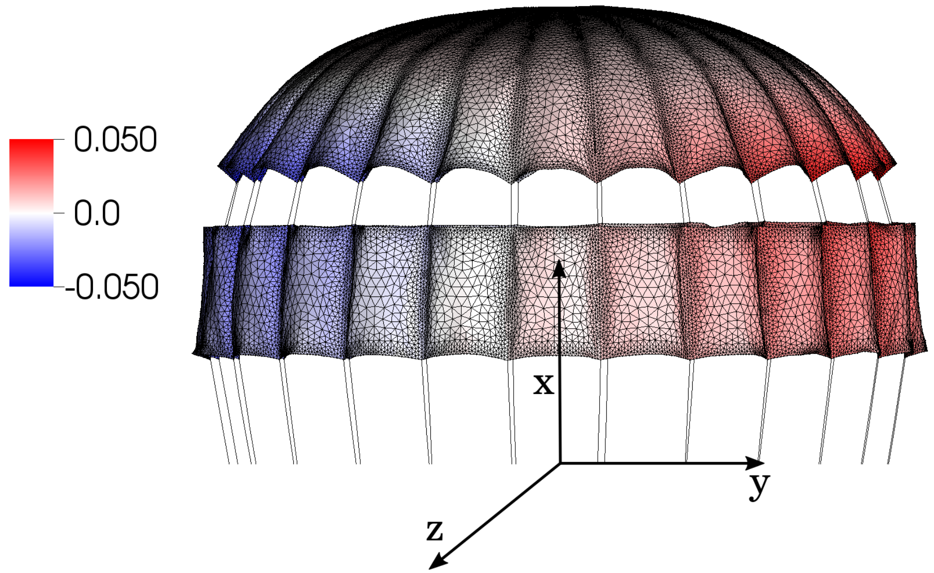


Simulation initial condition

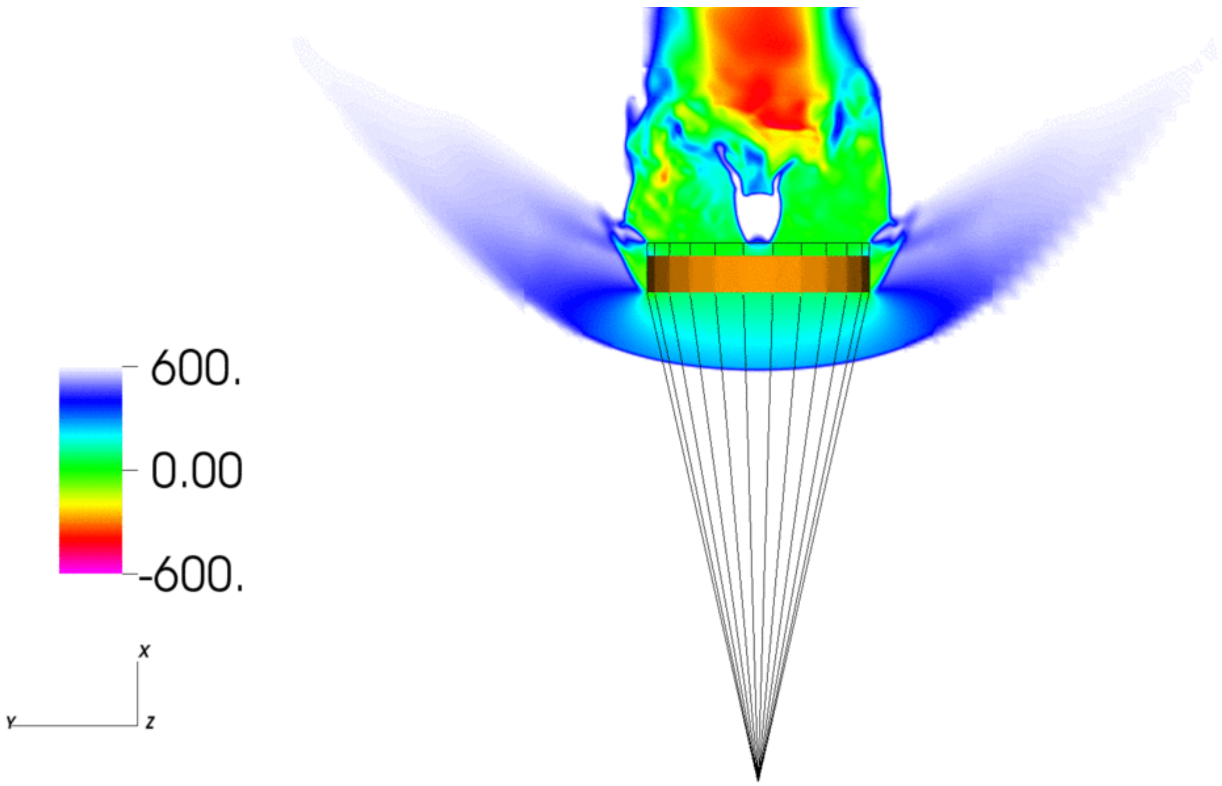
- 108,000 geometrically nonlinear shell elements resolve the disk and canopy

Problem Setup

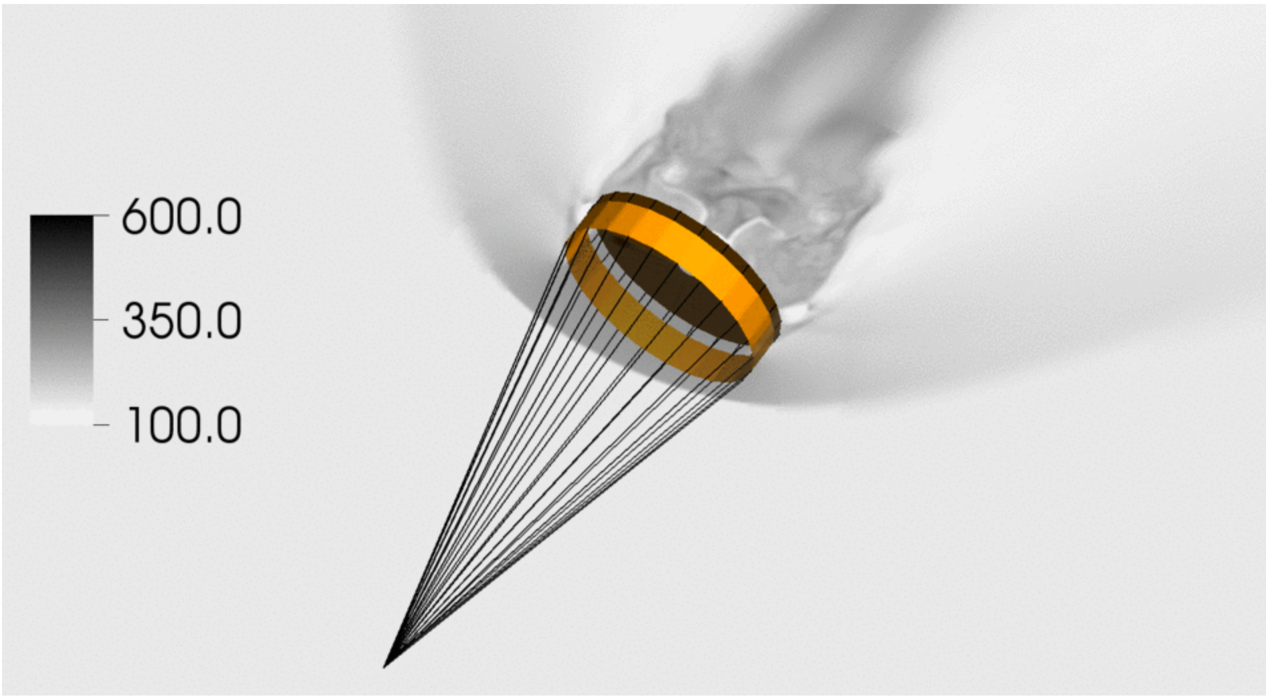
- Structural mesh based off simulations by Derkevorkian *et al.* (AIAA 2019)
  - Elements along seams are thickened by a factor of 4 to represent the stitching pattern used in manufacturing of the canopy
  - Finely resolving these regions also helps capture the stress discontinuities across the seams



Case 1: Uniform Flow (no capsule)



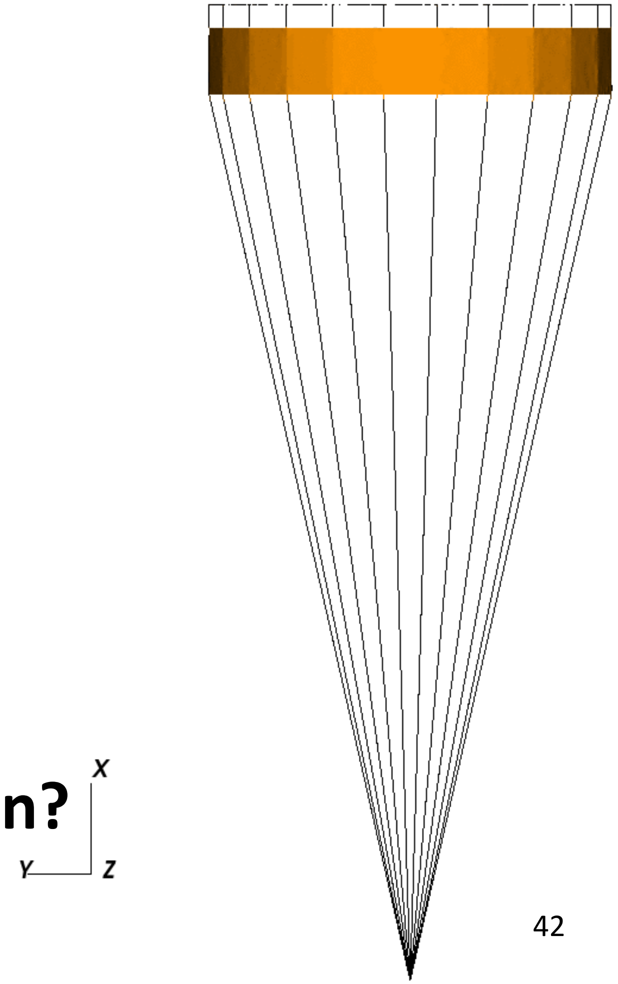
Streamwise velocity



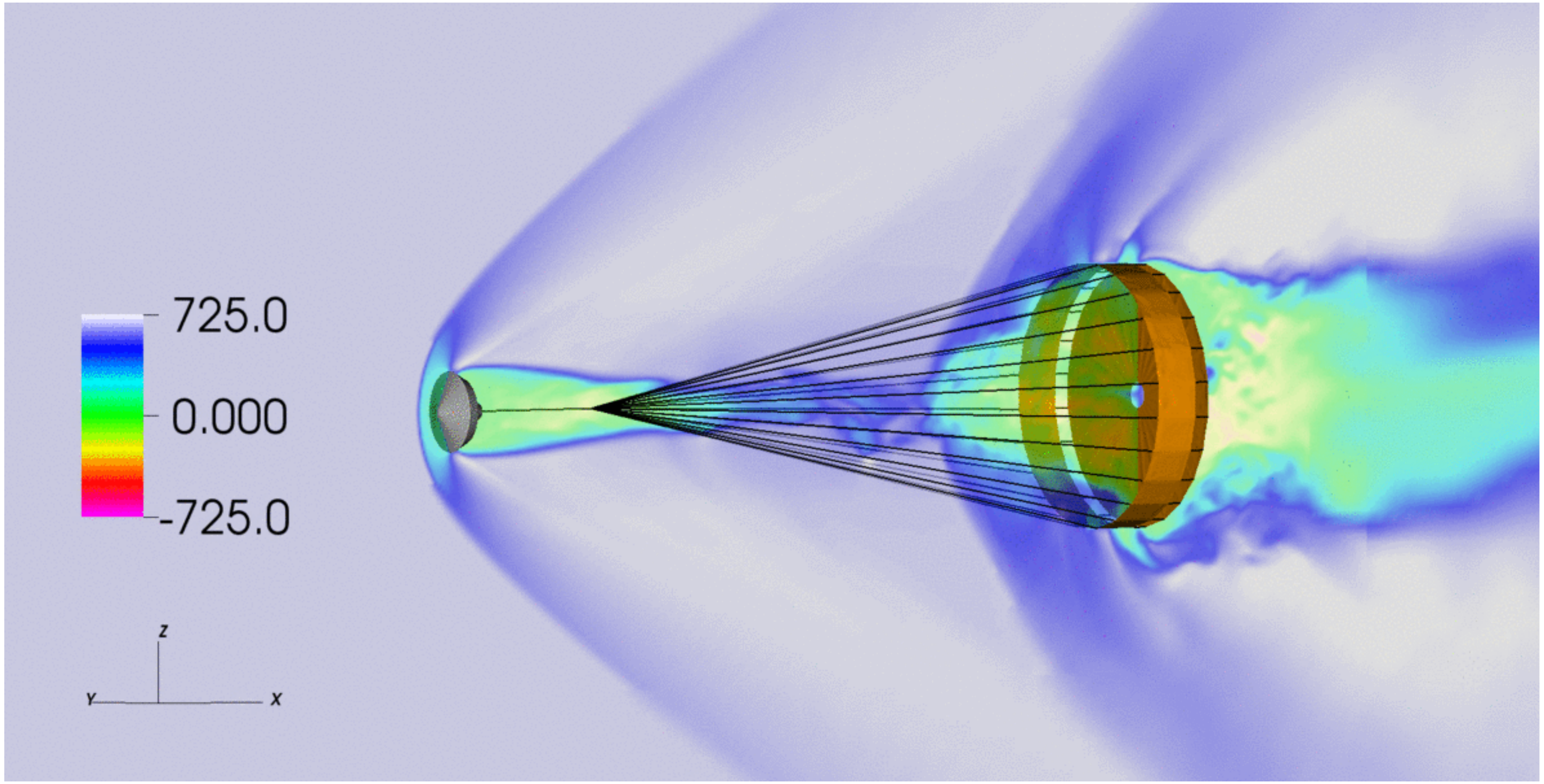
Temperature field

Case 1: Uniform Flow (no capsule)

- ❑ The cables are not resolved in the CFD volume mesh
  - o Nor do they experience any external loading → motion is virtually unopposed
  - o This leads to large period, large amplitude swaying of the cables
- ❑ The cables, as well as the canopy, start the simulation in an unstressed state
  - o There is no tension in the cables
- ❑ **Resolve with phantom geometry or approximate ling drag from damping matrix, reduced order model, *etc.*? Pre-tension?**



Case 2: Leading Viking-type Capsule



Streamwise velocity

Outline

- ❑ **Motivation/Introduction**
  - Mars, EDL system qualification, Simulation Capabilities
- ❑ **FSI Method**
  - Governing equations
  - Immersed Boundary Method for the Compressible Navier-Stokes Equations (CFD)
  - Geometrically Nonlinear Computational Structural Dynamics Solver (CSD)
  - Coupling procedure
- ❑ **Extended Validation for Fluid-Structure Interaction Problems**
- ❑ **Methods for Large-scale, Parallel CFD-CSD Coupling**
  - Disparate domain decomposition
- ❑ **Supersonic Parachute Inflation**
- ❑ **Summary and Outlook**

Summary and Outlook

❑ Summary:

- A validated method for FSI problems involving the large deformations of thin structures was extended to large, parallel simulations in supersonic flows

- The details of the weak, parallel coupling algorithm and the treatment of dealing with the disparate partitions in the CFD and CSD solvers were discussed

- The FSI method was then applied to two more large deformation FSI validation test cases to add onto the validation cases presented at SciTech 2019

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

Summary and Outlook

❑ **Outlook:**

- Treatment of the cable dynamics via damping, line drag
- Apply porous material boundary conditions on the canopy
- Implement more efficient contact algorithms for robustness
- Develop communication rings in the partitioned CFD-CSD solution procedure
- Reduce overhead and general optimization, load balancing

Questions?