STANDALONE SIMULATIONS OF SIAD AEROELASTIC RESPONSE USING LS-DYNA. R. A. Diaz-Silva[1] and N. Sarigul-Klijn[1], [1]Mechanical and Aerospace Engineering Department, University of California, Davis, CA, 95616, radiazs@ucdavis.edu

Introduction: Aeroelastic, three-dimensional simulations of a Supersonic Inflatable Aerodynamic Decelerator (SIAD) featuring a tension cone geometry are showcased in this presentation. This work is a follow-on to the authors' IPPW-9 poster [1] that studied SIAD inflation from a packed configuration without aerodynamic loads and a proof of concept quarter volume fluid-structure interaction (FSI) simulation. Partial inflation and buckling behavior results are now presented, together with a discussion of the simulation framework employed; its advantages and drawbacks.

FSI Simulations: Starting with the 980 beta release and later with the R7 version, a compressible flow solver is available in LS-DYNA (Livermore Software Technology Corporation). This solver allows both fluid-only simulations in a similar fashion as dedicated CFD software, but more interestingly, also makes it possible to perform multi-physics analysis in conjunction with the finite element method (FEM) structural solver. This capability enables FSI simulations in supersonic flow for structures that benefit from the proven accuracy of LS-DYNA.

In the work by Tanner et al. [2], multiple verification analyses were run with membrane shell elements including the same tension cone geometry presented here. The cited authors included fabric material modeling that is replicated for the current research. Bopp and Ruffin [3], further validated the use of LS-DYNA and presented FSI simulations by coupling the structural solver with an external CFD code.

The FSI simulations presented are standalone in the sense that the loose coupling between CFD and FEM occurs within the same program. Usually in a FSI simulation, a mapping operation must be performed between timesteps in which the aerodynamic and structural grids exchange data. This allows for the usually mismatching meshes to exchange displacements and deformed geometry for remeshing, together with pressure and the resulting nodal forces. In our approach, the structural mesh overlaps the fluid. This simplifies problem setup and solution time greatly but at the same time, it incorporates disadvantages such as the ability of the fluid to penetrate and pass through the structure for some mismatched meshes.

The method described corresponds to an immersed boundary type with a *direct-forcing, pulsing ghost-fluid approach strategy* [4]. Futhermore, fluid is solved by using the conservation element, solution element method (CESE).

The three-dimensional tension cone simulations are compared with the exhaustive work of Clark et al. [5] an their experimental wind tunnel inflation pressure sweep runs.

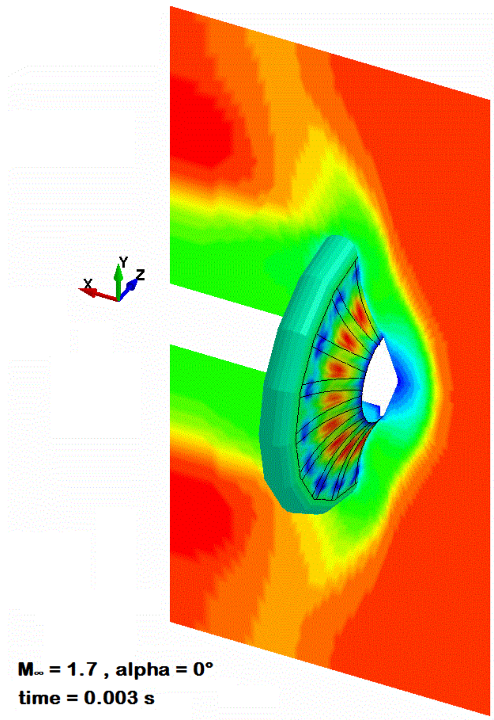

Figure: preliminary results for tension cone displacement. Cross-section at t = 0.003 s after rest state initial condition showing Mach number. Inviscid flow.

References: [1] Diaz-Silva R. A. and Sarigul-Klijn N. (2012) *IPPW-9*, Poster session. [2] Tanner C. L., Cruz and Braun (2010) *51st Struct., Struc. Dynamics, and Mat. Conf.*, AIAA 2010-2830. [3] Bopp M. S. and Ruffin S. M. (2013) *51st Aero Science*, AIAA 2013-0057. [4] Cook Jr. G. O., Zhang Z. and Im K. (2013) *21st CFD Conf.*, AIAA 2013-3070. [5] Clark I. G., Cruz, Hughes, Ware, Madlangbayan and Braun (2009) *20th Aero. Dec. Systems Tech. Conf. and Sem*, AIAA 2009-2967.