

Roles of CFD Simulations in Developing Rocket Propulsion System

Chongam Kim

Department of Aerospace Engineering,
Seoul National Univ. Seoul 151-744, Korea
chongam@snu.ac.kr

ABSTRACT

This paper focuses on the role of CFD in developing rocket propulsion system by simulating major devices such as turbopump inducer, cryogenic storage tank and solid rocket propellant. These are closely related to operation reliability and fuel efficiency of rocket propulsions system. The numerical computations on these devices have several issues owing to complex flow physics such as interaction between fluid, structure, and combustion domain or extreme flow conditions. This paper introduces these issues and corresponding numerical methods for more realistic simulation. Finally, several numerical results are presented to show the contributable aspects of CFD in rocket development stage.

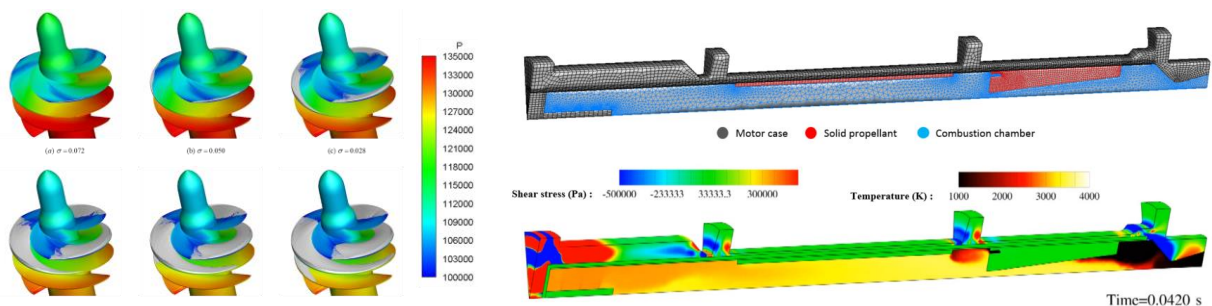


Fig 1. Example of numerical results on turbopump inducer(left) and solid rocket propellant(right)

REFERENCES

- Weiss JM, Smith WA. (1995) "Preconditioning applied to variable and constant density flows", *AIAA J.* **33**(11), 2050-2057
- Hyeongjun Kim, Daeho Min, C. Kim(2014) "Computations of Cryogenic Cavitating Flows around Turbopump Inducer", *Proceedings of ICCFD8*, Cheongdu, China
- Donea, J., Huerta, A., Ponthot, J., & Rodr, a. (1999). "Arbitrary Lagrangian -Eulerian Methods." *Encyclopedia of Computational Mechanics*.
- Sangho Han, Chongam Kim(2014), "Integrated Fluid-Structure Simulation for Full Burning of a Solid-Propellant Rocket Interior", *J. Propulsion and Power*, **30**, 883-900