

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 propulsion 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.

1. INTRODUCTION

The rocket systems for space exploration are composed of payload, guidance, propulsion system and structural assemblies. Among these elements, increasing the portion of payload has been on the rise as a recent major issue, because the mission of rocket system depends on payload itself. But most portion in entire weight of rocket is occupied by liquid propellant, and consequently, reducing structure weight of fuel tank or estimating optimum amount of fuel are demanded to save the redundant weight. In this sense, liquid propulsion part of rocket system generally employs the turbopump inducer to omit unnecessary structure required for highly compressed cryogenics. Furthermore, many research institutes are concentrating on precise prediction of remaining fuel in tank to achieve the high fuel efficiency.

At the same time, reliable launching should be secured to prevent enormous costs as well as tragic accidents by operation failure. It has been considered as a first fundamental issue to be assured for rocket propulsion system, but operation failures are occasionally occurred even in recent days. However, triggering reasons for failure are generally diverse and complex for examination. For example, failure of H-II rocket has come from vibration stress due to swirling cavitation in turbopump, while explosion of Titan IV rocket has come from bore choking resulting from excessive deformation of

solid propellant.

Even though many other issues still exist to be resolved, this paper focuses on aforementioned topics, operation reliability and fuel efficiency. As stated above, turbopump inducer, which is employed to reduce the weight of fuel feed system for efficiency, is closely related to operation reliability because of cavitation. The cavitation is commonly known as inevitable phenomena that phase is changed locally from liquid to gas phase by pressure decrement around rapid rotating blade. This generated cavitation impedes not only suction performance of inducer, but also stability of propulsion system as H-II rocket or Ariane V failure. Therefore, understanding the characteristics of cavitation and its effects on suction performance or operation stability are crucial in design process of propulsion system.

As the next problem, the behavior of cryogenic fluids in microgravity condition is handled correlating fuel efficiency. The remainder can be easily measured in normal gravity, but microgravity in space makes difficult to measure exactly because it obscures the interface between liquid and gas phase. Furthermore, various heat sources, which cryogenic storage tank may experience in space, make a loss in liquid phase by ebullition. In this sense, predicting optimum amounts of cryogenic fluids by considering ebullition loss and its change due to behavior of fluids in microgravity have important meaning in development stage of rocket propulsion system.

Finally, complex multi physical phenomena inside solid rocket is treated to avoid catastrophic accidents by rocket failure. The burning process of propellant grain generates high pressure and temperature gases, which gives a structure load and following deformation on to the grain. Additionally, it decreases the volume of propellant and changes the surface shape as well as fluid domain region. This change reversely causes burning rate and pressure distribution on the propellant surface.

Examining these problems can be conducted through experimental manner, but various limitations make difficult to accomplish the objectives. For example, high pressure and temperature condition in solid rocket propellant hinder visualizing inside rocket motor. Furthermore, reproducing various gravity or heat source condition similar to real mission through long period of time is also difficult. Recently, numerical approach is gathering the attention as an alternative to experimental research. It can help realizing local flow physics comparably in ease than experimental research, while it requires lower cost. The extreme flow conditions, which were difficult to reproduce in experimental manner, also can be simply implemented in numerical approach. Furthermore, recent growth in computing power makes numerical research more attractive.

Based on this background, cavitating flows around turbopump inducer and behavior of cryogenic fluids in microgravity condition are numerically simulated in multiphase framework, while multidisciplinary physics inside solid rocket is computed by integrating fluid, structure, and combustion domain. The numerical issues and corresponding methods are introduced firstly. Then, simple validation and some examples, which can show the role of CFD in developing rocket propulsion system, are presented.

2. Cryogenic multiphase flows in liquid rockets

There are diverse numerical issues that must be resolved for the calculation of cryogenic multiphase flows. We categorize the issues into a group regarding multiphase flows and cryogenic flows. After addressing the numerical issues and the way we overcome each difficulty, several numerical results will be shown: flows inside the fuel tank and cavitation around turbo-pump inducer.

2.1 Governing equations

The homogeneous mixture equations are adopted to describe cryogenic multiphase flows. We assume the mixture of two different phases as a pseudo-fluid whose properties are determined by the suitable averages of the constituent phases. Since the pseudo-fluid is not necessarily an equilibrium state of liquid and gas phases, one phase mass conservation law is additionally needed to account for the phase change. Thus, the governing equations consist of mixture mass, momentum, and energy conservation laws with one phase mass conservation law. In order to include the non-condensable gas phase, one more mass conservation equation is added, resulting in equation (1) as our governing equations.

$$\Gamma \frac{\partial}{\partial \tau} \int_{\Omega} Q d\Omega + \frac{\partial}{\partial t} \int_{\Omega} W d\Omega + \oint_{\partial\Omega} [F - F_v] dS = \int_{\Omega} T_{pc} d\Omega \quad (1)$$

In equation (1), Q represents the vector of primitive variables as

$$Q = [p \quad u \quad v_w \quad T \quad y_v \quad y_g], \quad (2)$$

where y_v and y_g stand for the mass fraction of vapor and non-condensable gas phase, respectively. The mixture density and enthalpy are defined following Amagat's law as equation (3):

$$\frac{1}{\rho} = \frac{1 - y_v - y_g}{\rho_l(p, T)} + \frac{y_v}{\rho_v(p, T)} + \frac{y_g}{\rho_g(p, T)}$$

$$h = h_l(p, T)[1 - y_v - y_g] + h_v(p, T)y_v + h_g(p, T)y_g \quad (3)$$

2.2 Multiphase flows

To simulate multiphase flows, various problems should be overcome. In this paper, we categorized these problems according to three issues. The first issue is 'extension to all-speed flow regime'. Second one is 'Capturing of shock/phase discontinuity'. The last one is 'phase change phenomena'. From now on the method which is used to treat these issues will be accounted for.

2.2.1 Numerical issue 1: Extension to all-speed flow regime

When we use compressible flow solver to compute the governing equation addressed, we could be faced with stiffness problem in low-Mach number flow. That is because the acoustic speed varies drastically in mixture region. Therefore, this problem can lead to convergence problem. To achieve reasonable results, it need to alleviate

the system stiffness for all-speed flows. There are two methods which we used at the same time to reduce the system stiffness.

The first method is system preconditioning technique of Weiss and Smith[1]. This method substitutes eigenvalues making the system stiffened as below.

$$\lambda \left(\Gamma^{-1} \frac{\partial F}{\partial Q} \right) = U, U, U, U, U, U', U' + D, U' - D \quad (4)$$

$$\text{where, } U' = \frac{1}{2} \left(1 + \frac{c'^2}{c^2} \right) U, D = \frac{1}{2} \sqrt{\left(1 - \frac{c'^2}{c^2} \right)^2 U^2 + 4c'^2}$$

When the method is applied, the condition number of system matrix can be controlled by defining preconditioned speed of sound(c'). so, we can make condition number of system matrix approximately order of one as mentioned below.

$$c' = \min(c, \max(V, V_{CO}, V_{un})) \quad (5)$$

$$\text{where, } V_{un} = \frac{L}{\pi \Delta t}$$

In equation (5), the c , V , V_{CO} and V_{un} are physical speed of sound, local flow velocity, cut-off velocity and unsteady preconditioning velocity, respectively. We customarily regard cut-off velocity as referential velocity or free stream velocity of problems. In case of unsteady preconditioning velocity, it has relevance to Strouhal number which represents frequency of problems. So, we calculate this velocity with characteristic length scale that is taken as the problem domain size and a representative time scale from the lowest wave.

The second method is scaling of numerical dissipation. The flux schemes we used calculate numerical dissipation with Mach number. However, the numerical dissipation could be inaccurate in low-Mach flows because of relatively huge acoustic speed. In this reason, we should apply proper scaling for velocity-difference term and pressure-difference term respectively for correct numerical dissipation. The scaling functions proposed by Kim *et al.*[2] in steady and unsteady low-Mach number flows are accounted for as below.

$$\phi_p = \theta_p (2 - \theta_p) \quad , \quad \theta_p = \min \left(1, \max \left(\frac{\sqrt{u_{1/2}^2 + v_{1/2}^2 + w_{1/2}^2}}{c_{1/2}}, \frac{V_{CO}}{c_{1/2}}, \frac{V_{un}}{c_{1/2}} \right) \right) \quad (6)$$

for pressure difference term considering unsteady preconditioning parameter.

$$\phi_v = \theta_v (2 - \theta_v) \quad , \quad \theta_v = \min \left(1, \frac{\sqrt{u_{1/2}^2 + v_{1/2}^2 + w_{1/2}^2}}{c_{1/2}} \right) \quad (7)$$

for velocity difference term.

2.2.2 Numerical issue 2: Capturing of shock/phase discontinuity

The main feature of RoeM or AUSMPW+ is adjusting numerical dissipation near the shock by employing shock discontinuity sensing term. It was defined with only static pressure in single phase computation. However, the shock discontinuity sensing term which introduced in RoeM or AUSMPW+ may have a problem in multiphase flows. Especially, near liquid region, pressure changes drastically even without shock due to large disparity of density between liquid and gas phase. And it can lead to misinterpret non-shock flow regions as shock flow regions. To overcome this difficulty, we introduced new shock discontinuity sensing term in multiphase flows considering density and speed of sound as well as static pressure according to previous researches[2].

$$\Pi_{1/2} = \min\left(\frac{\dot{p}_L}{\dot{p}_R}, \frac{\dot{p}_L}{\dot{p}_R}\right) \quad , \quad \dot{p}_{L,R} = P_{L,R} + \min(\rho_L, \rho_R)c_{1/2}^2 \quad (8)$$

2.2.3 Numerical issue 3: Phase change phenomena

The phase change model can be categorized into pressure driven process, which is cavitation models and temperature driven process, which is ebullition models. These models are driven in non-equilibrium thermodynamic phenomena. For cavitation models, mass changing rate between phases is proportional to difference between local pressure and saturation pressure. The representative examples of such models are Merkle[3], Kunz[4], FCM[5], and Schnerr-Sauer[6] model. In case of ebullition models which is represented to Lee's model[7], the mass changing rate is proportional to difference between local temperature and saturation temperature.

2.2.4 Result 1: Unsteady cavitation around 2-D wedge

One of famous multiphase flow problems is unsteady cavitation around 2-D wedge. This problem simulates relatively low speed water flows 2-D wedge and cavities occurred behind wedge is observed. Our research team experimented this problem as well. So, we can compare both of results and validate our multiphase solver.

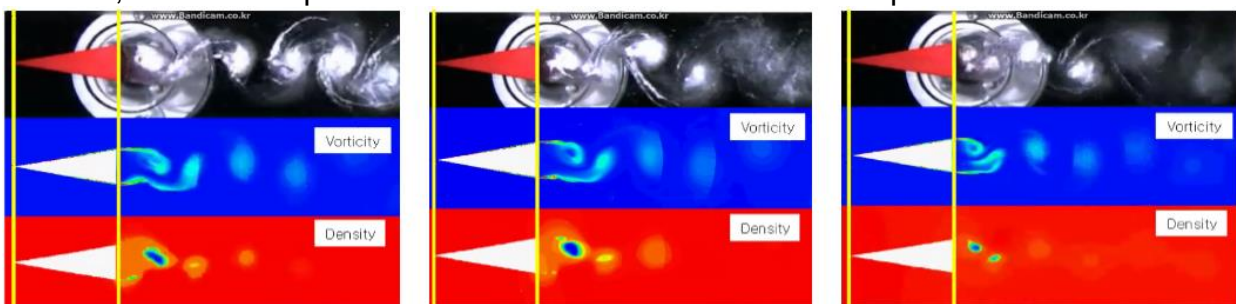


Figure 1. σ (cavitation number) = 1.24 (left), 1.48 (middle), 2.05(right)

Figure 1 present comparison between numerical results and experimental data in various cavitation numbers. We can see that the locations of vortices or low density regions have good agreements with the shedding patterns from experimental data. So, we can conclude that the numerical model we proposed show similar behaviors of experiment and our multiphase model is validated. It can be certain when from the FFT

analysis of both results as Figure 2, which presents the dominant frequencies of free shedding vortex.

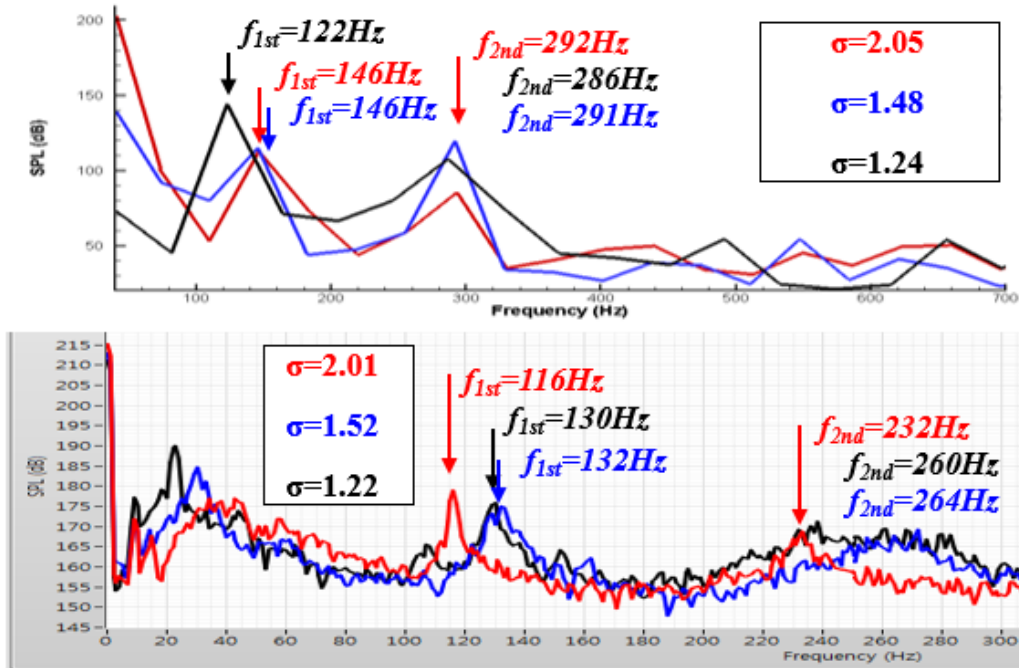


Figure 2. FFT analysis (upper: CFD, lower: Exp.)

2.2.5 Result 2: 2-D shock wave-water column interaction

Another famous validation case of multiphase flows is 2-D shock wave-water column interaction. It is a severe benchmark test for compressible multiphase flow because it need to capture the correct shock locations. This problem simulates that moving air shock whose speed is 1.47 Mach number strikes on a water-column. After striking, there are several complex shocks and expansions which can be seen as Figure 3. And it also has good agreement with results of other research groups[8].

2.3 Cryogenic flow

2.3.1 Numerical issue 1: Reflecting the real fluid properties

Since most cryogenics are working near their critical points, thermal effects play an important role. For example, when a cavitation occurs, the temperature drops more due to the low density ratio between liquid and gas phases near the critical temperature. Consequently, additional cavitation is suppressed as the saturation pressure is decreased. These thermodynamic effects with phase change are all entangled with real fluid properties.

Therefore, exact equation of state such as MEOS or NIST database [9~11] for specific working fluid is essential for an accurate computation. Employing these equations directly, however, requires huge computational costs. We use tabular function with database hashing based on NIST as an equation of state for subcooled,

superheated, and supercritical states for the computational efficiency. SRK equation of state is also applied for non-condensable gases in cryogenic tank simulation.

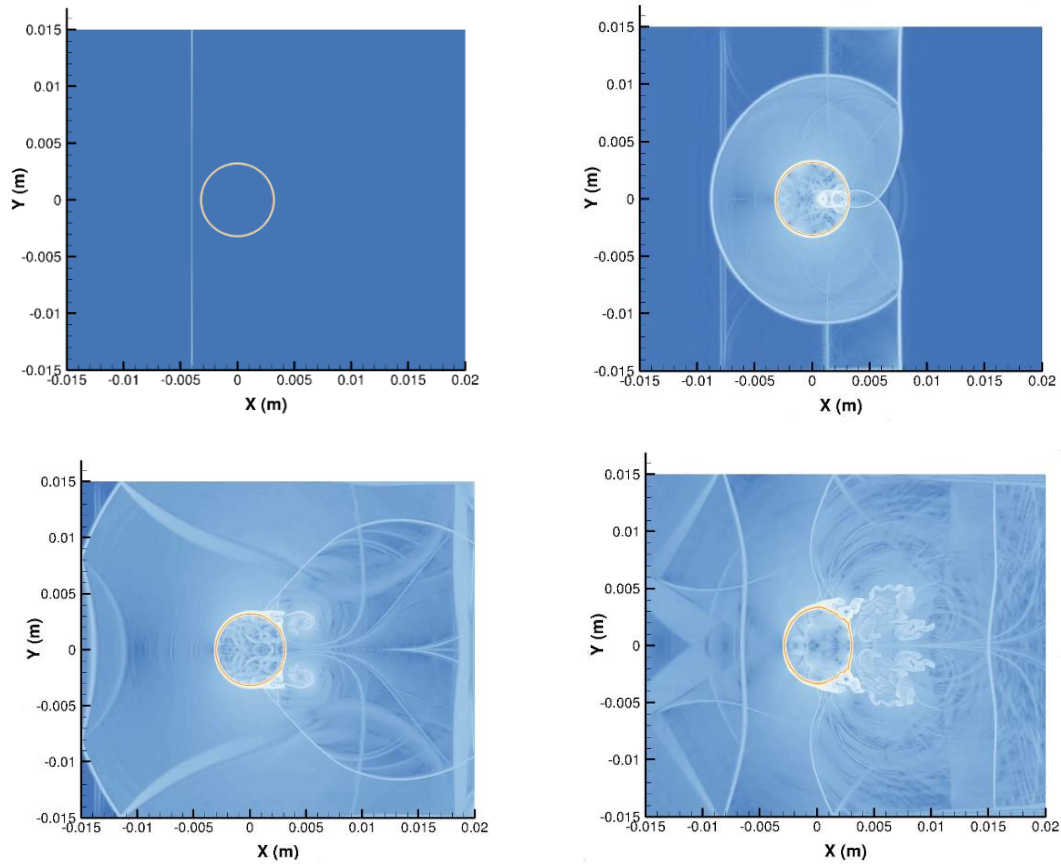


Figure 3. Numerical Schlieren of shock/water-column interaction, $t=0.0 \mu\text{s}$ (upper, left) , $t=3.0 \mu\text{s}$ (upper, left) , $t=7.0 \mu\text{s}$ (upper, left) , $t=100.0 \mu\text{s}$ (upper, left)

2.3.2 Result 1: Cryogenic cavitating flow around hydrofoil

As a validation case, numerical simulations of experiments by Hord [12] for liquid nitrogen is presented. This cryogenic cavitating flow is known to exhibit substantial temperature variation, hence the importance of accurate equation of state is emphasized.

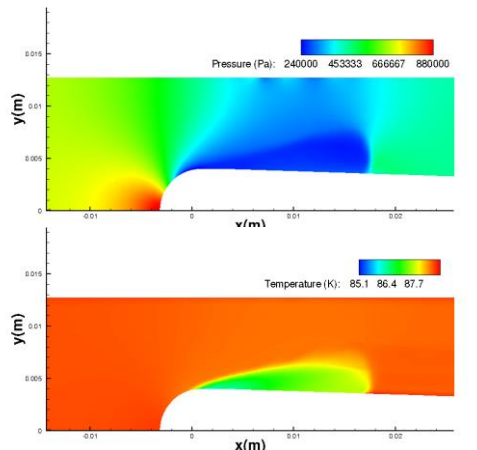


Figure 4. Pressure and temperature contours, ASDL

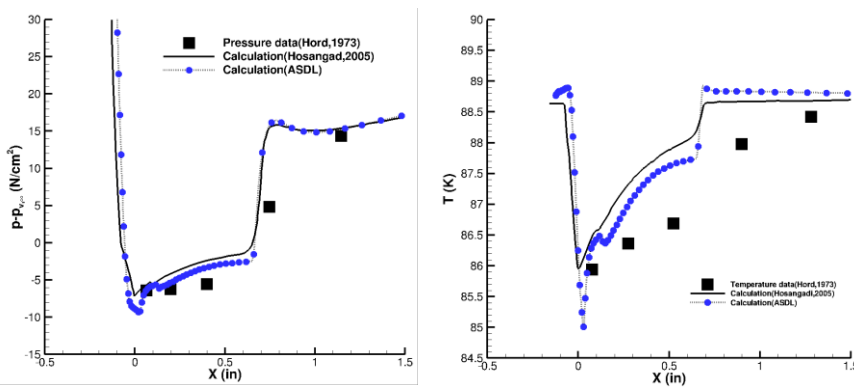


Figure 5. Pressure depression and temperature distribution, ASDL

Figure 4 clearly shows the mushy characteristics of cryogenic cavitation. Thermal effect is successfully reflected and the cavitation length is well predicted as in Figure 5. From the results, we can find the use of hash function based on NIST database is appropriate for the computation of cryogenic multiphase flows.

2.4 Flows inside the propellant tank

Figure 6 represents pressurizing process of liquid nitrogen storage tank under normal gravity and microgravity circumstance. The injected hot helium gas approaches the liquid nitrogen surface triggering the boiling process in microgravity condition, while the helium is floating on the upper part of the tank due to its small density under normal gravity. The amount of loss in liquid fuel under microgravity situation can be considerable due to this active boiling. In preparing stage of vehicle launch, these fuel loss should be taken into account for the successful and efficient mission.

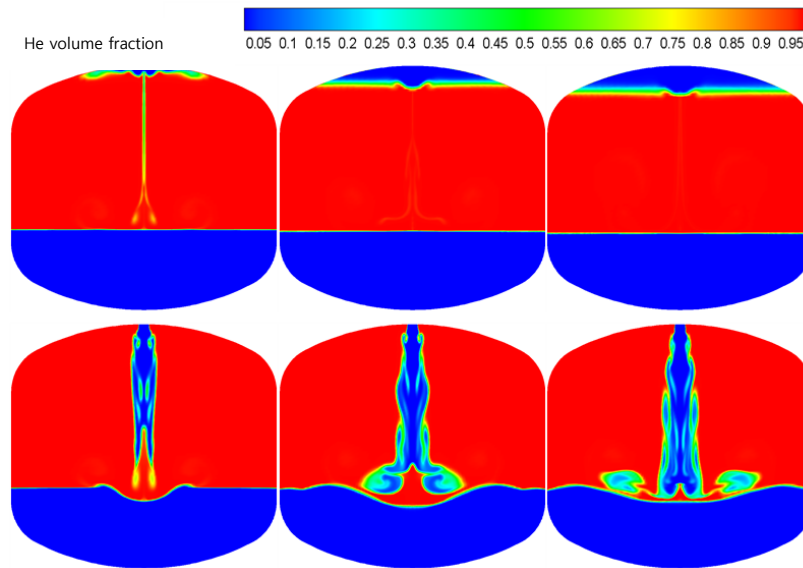


Figure 6. Volume fraction evolution of helium injection under normal gravity (upper) and microgravity (lower), ASDL

Figure 7 shows another active pressurization in liquid oxygen tank without gravity. The tank is located at NASA SSC E-1 test complex. We simulate the injection of supercritical oxygen. Pressure rises quickly due to the high mass flow rate as in figure 7 (f). Therefore, condensation is the major phase change phenomenon in this case. We can see vigorous mixing in ullage and consequent deformation of phase interface. As seen in these example, numerical approach on cryogenic flow containing phase changing process can produce data for efficient management of fuel or oxidizer.

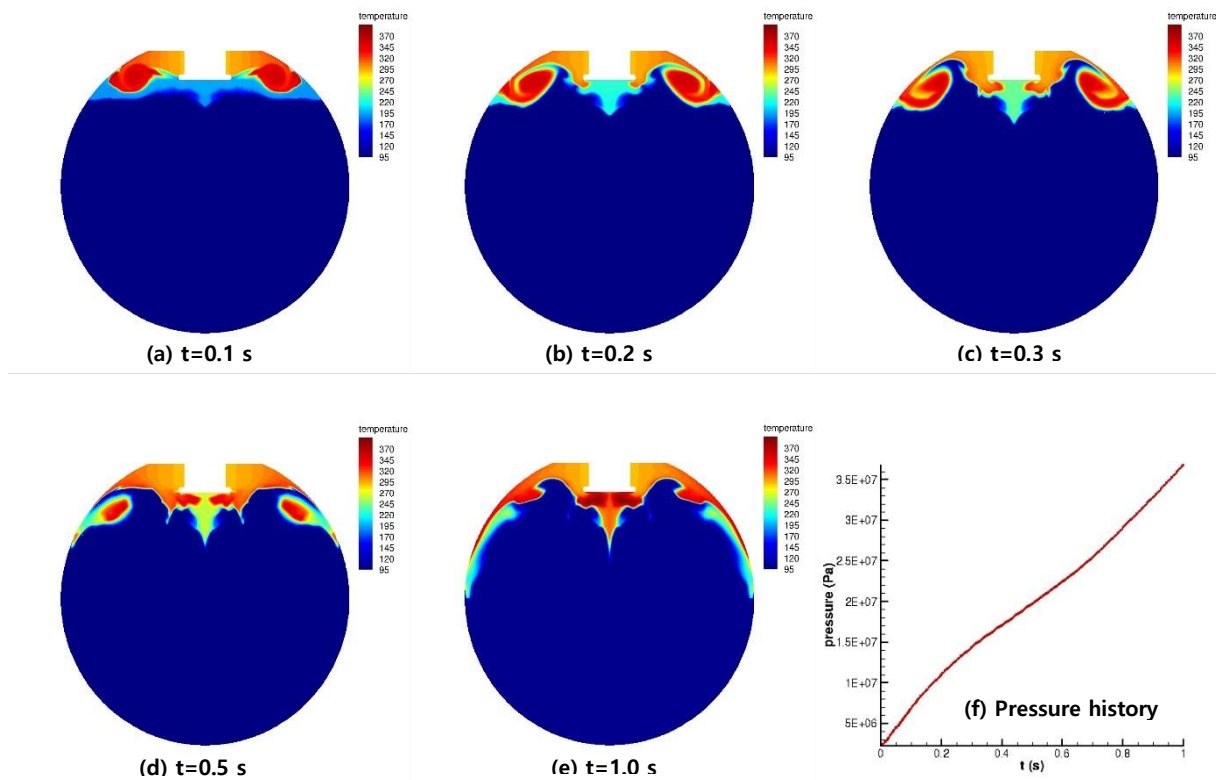


Figure 7. Temperature distribution along time for E-1 tank, ASDL

2.5 Cavitation around turbo-pump inducer

Cavitating flows around turbopump inducer are numerically reproduced. The target inducer is experimented by KARI. Figure 8 compares the experimentally measured pressure and suction performance of inducer with our computed results. It indicates that our numerical methods can predicts its suction performance reasonably, and hence it can be used as a substitution for experimental works. Furthermore, Figure 9 which represents surface pressure distribution and cavitation region implies that our numerical approach can help realizing characteristics of cryogenic cavitation. Further researches on unsteady physics are expected to be valuable for operation reliability.

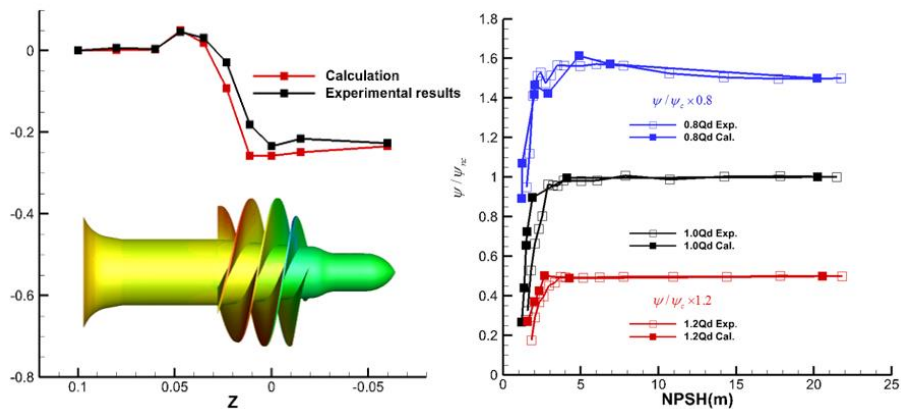


Figure 8. Comparison between measured experimental data and computed result in non-cavitating region (left), predicted headrise coefficient (right).

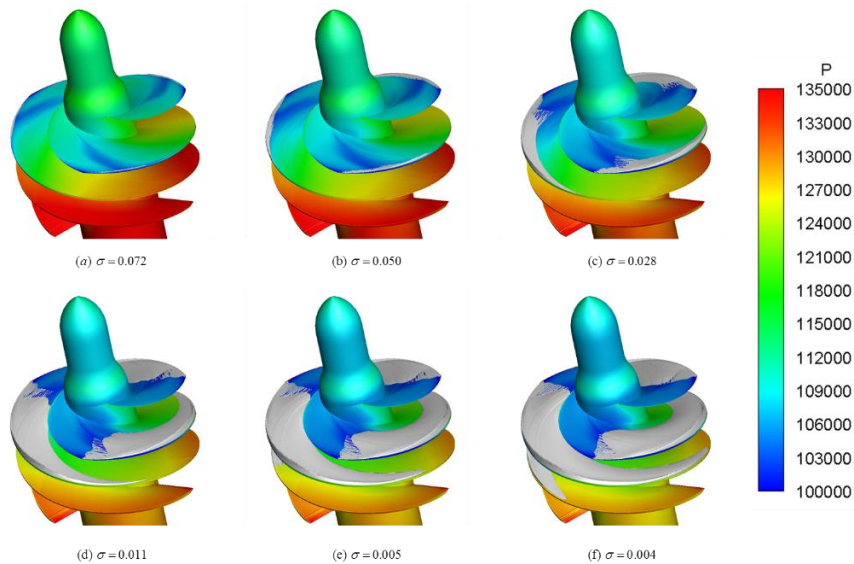


Figure 9. Pressure distribution on the inducer surface and cavity region in liquid hydrogen.

3. Multidisciplinary physics inside solid rocket

The numerical issues related to the solid rocket simulation can be categorized into two groups, domain deformation by the propellant burning process and methods for coupling fluid, structure and burning phenomena. Diverse methods are implemented to overcome each issue. With these methods, integrated Fluid-Structure-Burning Interaction (FSBI) simulations are performed to analyze multidisciplinary physics inside solid rocket.

3.1 Governing equation

A governing equation should be modified to describe the deformation of mesh. Arbitrary Lagrangian–Eulerian (ALE) description [13] determines the appropriate relationship between the continuum and the deforming mesh of computing domains.

$$\frac{\partial}{\partial t} \int_{\Omega(t)} U dV + \int_{\partial\Omega(t)} (F(U) - \dot{x}U) \cdot \dot{n} dS + \int_{\partial\Omega(t)} (G(U) \cdot \dot{n}) dS = 0$$

$$U = [\rho, \rho u, \rho v, \rho w, \rho E]$$

3.2 Domain deformation

3.2.1 Numerical issue 1: Geometric Conservation Laws (GCL) condition

In ALE description, face velocity is required for evaluating convective flux. Geometric Conservation Law (GCL) condition [13] defines face velocity based on geometric information. GCL states that the change in grid volume must be equal to the volume swept by grid boundary. Severe degradation in overall simulation accuracy occurs unless GCL condition is considered.

3.2.2 Numerical issue 2: Mesh moving method

After the ignition, both fluid and structural domains are deformed by propellant burning process. As the domain boundary moves, mesh distribution of the internal domain should be deformed appropriately to maintain mesh quality. For fast and robust mesh moving, Inverse Distance Weighting (IDW) method [14] is adopted.

$$w(x) = \frac{\sum_{i=1}^n v_i \phi(r_i)}{\sum_{i=1}^n \phi(r_i)}, \phi(r) = r^{-c}, r_i = \|x - x_i\| \geq 0$$

n : the number of boundary node, x_i : boundary node, x : interior node
 v_i : boundary node displacement, w : interior node displacement

3.2.3 Numerical issue 3: Automatic re-meshing process

The decrease of the mesh quality is occurred inevitably by continuous mesh moving. In order to maintain the quality, an automatic re-meshing process is implemented. If the mesh quality drops below certain criterion, re-meshing is conducted using Delaunay triangulation.

3.2.4 Numerical issue 4: Data transfer during re-meshing process

During re-meshing process, solution data must be transferred from old to new mesh. Consistent interpolation with quadtree / octree data structure is adopted for data transfer.

3.3 Coupling methods

3.3.1 Numerical issue 1: Propellant grain burn back simulation

Propellant grain burn back simulation requires a surface-tracking method for precise burning area prediction. Most popular surface-tracking method is Level Set Method (LSM). LSM traces surface in an implicit representation and treats topological changes easily. However, LSM suffers from high computational cost. Because efficiency is important in multidisciplinary physics simulation, it is almost impossible to implement LSM in our 3-D simulation.

Face Offsetting Method (FOM) [15] moves grid nodes and faces explicitly by assigned grid velocities. Although FOM cannot handle topological changes easily, this method can be used at low computational cost. FOM is adopted as our surface-tracking method for efficiency.

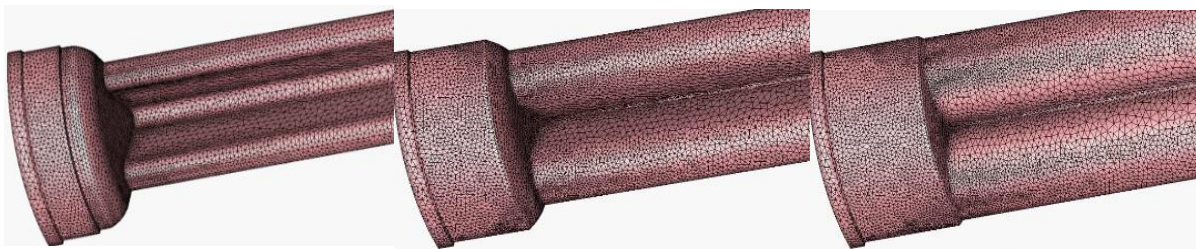


Fig. 1 Grain burn back simulation of Erosive Burning Motor (EBM), NCIA Lab. (SNU)

3.3.2 Numerical issue 2: Data transfer at non-matching interface

In general, grid element sizes for fluid are small compared to structure at the fluid-structure interface. Therefore, fluid and structure meshes are non-matched, and the ability to transfer data at this non-matching interfaces is required. Common-refinement method [16] using Sobolev minimization is one of the well-known data transfer method at non-matching interface. We use common-refinement method for 2-D simulation. In 3-D simulation, however, 3rd order interpolation method is adopted for efficiency.

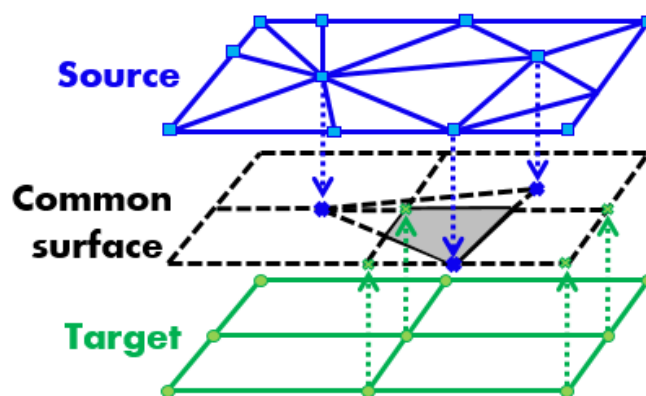


Fig. 2 Constructing common surface from source and target mesh

3.3.3 Numerical issue 3: Time stepping scheme for integrated simulation

In multidisciplinary physics, a time integration method combining individual simulations is needed. Accuracy and stability of the integrated simulation depend on the time integration scheme. Conventional Serial Staggered (CSS) method is the most popular method in Fluid-Structure Interaction (FSI) simulation. This method is successfully implemented for Fluid-Structure-Burning Interaction (FSBI) simulation.

3.4 Integrated FSBI simulation

Fig. 7 shows ignition delay and secondary ignition phenomena during combustion. [17] Initial cold gas and separation bubble make the ignition delay and secondary ignition occurs at propellant surface.

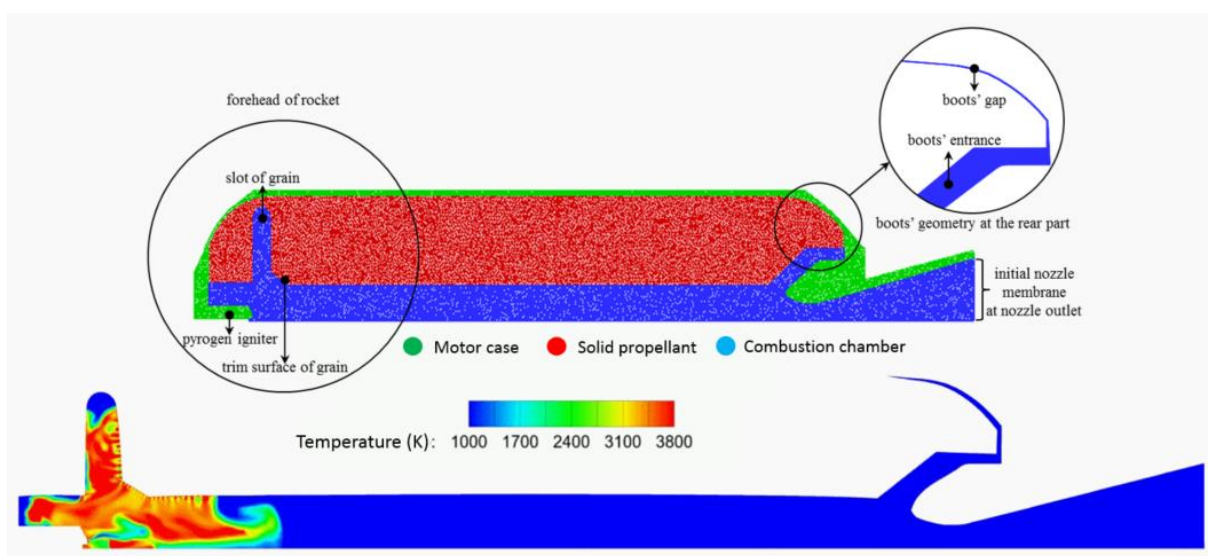


Fig. 3 Temperature contour

Fig. 8 represents an erosive burning effect in high L/D rocket model. Additional heat flux induced by high velocity core flow causes erosive burning effect on the propellant grain surface. This phenomenon is prominent as the flow approaches the nozzle exit.

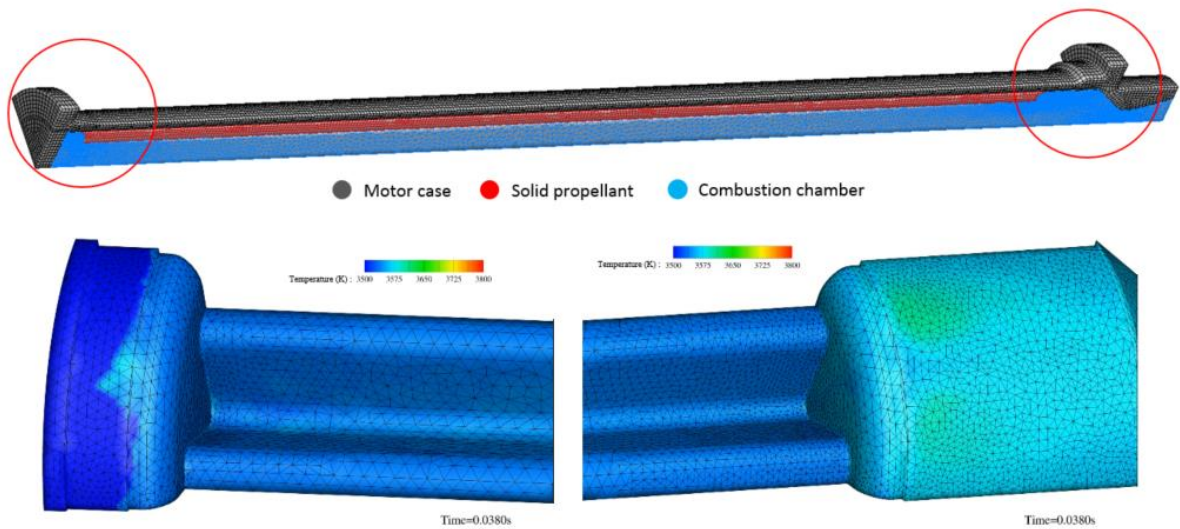


Fig. 4 Numerical result inside combustion chamber

Fig. 9 represents the failure mode similar to the explosion of Titan IV. Inward deformation of the aft propellant segment make the core gas flow choked at the aft segment.

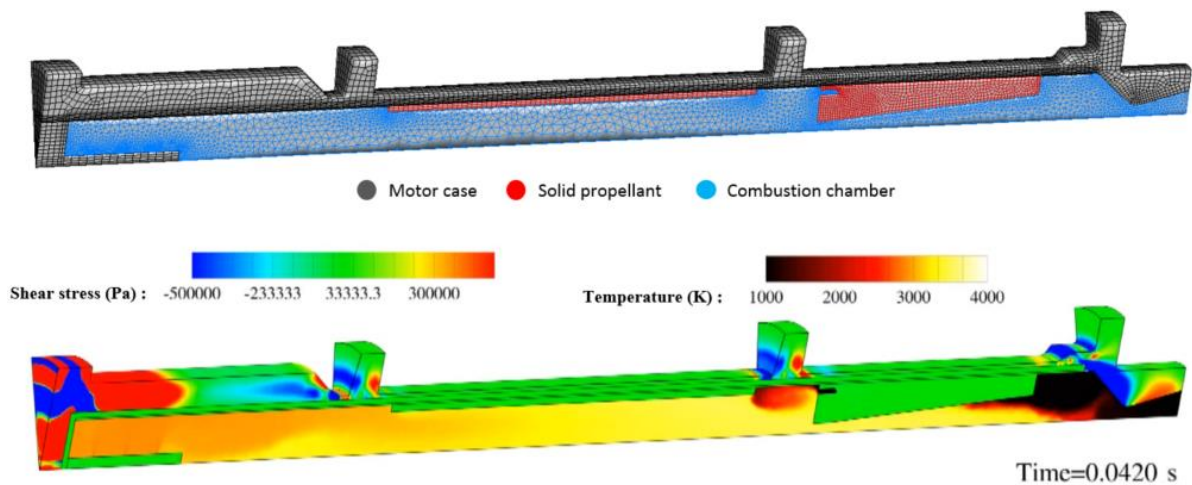


Fig. 5 Shear stress and temperature contour

4. Conclusion

The important physics related to operation reliability and fuel efficiency of rocket propulsion system are numerically studied in this paper. It includes cavitation around turbopump inducer, behavior of cryogenic fluids in microgravity condition, and multi physical phenomena around solid rocket propellant. There are some numerical issues which may make difficult to provide reliable results. This paper introduces these numerical issues and corresponding methods. Then, some numerical examples are successfully simulated.

The simulations on cavitation around turbopump inducer can help predicting cavitation breakdown point. Furthermore, it can provide valuable data to investigate unsteady cavitation instability characteristics. The capability predicting the remainder of fuel tank can be applied to maximize fuel efficiency. The CFD also can be used to understand complex unsteady physics inside solid rocket. It can predict unexpected failure mode of solid rocket by examining fluid field around rocket propellant, which is complicatedly coupled with structure and combustion domain, and consequently improve the reliability in early design stage.

By summarizing these applicable aspects, it can be concluded that CFD can contribute to the development and performance enhancement for rocket propulsion system.

Acknowledgements

The author appreciates the financial supports provided by National Space Laboratory program(NRF-2014M1A3A3A02034856) and Development of Space Core Technology program (NRF-2015M1A3A3A05027630) through the National Research Foundation of Korea funded by the Ministry of Science, ICT & Future Planning. The author also appreciates the help of Daeho Min, Hyeongjun Kim in preparation of this manuscript.

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
- Merkle, C. L., Feng, J.Z., Buelow, P.E.O., (1998), "Computational modeling of the dynamics of sheet cavitation", *Proceedings of 3rd International Symposium on Cavitation*, Grenoble, France
- Kunz, R.F., Boger, D.A., Stinebring, D.R., Chyczewski, T.S., Lindau, J.W., Gibeling, H. J., Venkateswaran, S. Govindan, T. R., (2000), "A preconditioned Navier-Stokes method for twophase flows with application to cavitation prediction", *Computers and Fluid*, **29**, 849-875
- Ashok K. Singhal, Mahesh M. Athavale, Huiying Li, Yu Jiang, (2002), "Mathematical Basis and Validation of the Full Cavitation Model", *J.Fluid Engineering*, **124**, 617-624
- G.H.Schnerr and J.Sauer, (2001), "Physical and numerical modeling of unsteady cavitation dynamics", *Proceedings of 4th international conference on multiphase flow*, New Orleans, USA
- W. H. Lee, (1979), "A Pressure Iteration Scheme for Two-Phase Modeling". Technical Report LA-UR 79-975. Los Alamos Scientific Laboratory, Los Alamos, New Mexico.
- K. Kitamura and M.-S. Liou, (2012), "Comparative study of AUSM-family schemes in compressible multiphase flow simulations", in *Proceedings of ICCFD7*, Big Island, Hi.
- Roland Span, Eric W. Lemmon, Richard T Jacobsen(2000), "A reference equation of state for the thermodynamic properties of nitrogen for temperature from 63.151 to 1000K and pressure to 2200Mpa", *J. Phys. Chem. Ref. Data*, **29**

The 2016 Structures Congress (Structures16)

Jeju Island, Korea, August 28-September 1, 2016

- Richard B. Stewart, Richard T. Jacobsen, W. Wagner(1991), "Thermodynamic properties of oxygen from the triple point to 300K with pressure to 80Mpa", *J. Phys. Chem. Ref. Data*, **20**
- J. W. Leachman, R. T. Jacobsen, S. G. Penocello, E. W. Lemmon(2009), "Fundamental equation of state for parahydrogen, normal hydrogen, and orthohydrogen", *J. Phys. Chem. Ref. Data*, **38**
- J. Hord, L. M. Anderson, W. J. Hall, "Cavitation in liquid cryogenics : 2-Hydrofoil", NASA Contraction Report : NASA CR-2054
- Donea, J., Huerta, A., Ponthot, J., & Rodr, a. (1999). "Arbitrary Lagrangian -Eulerian Methods." *Encyclopedia of Computational Mechanics*.
- Jeroen A.S. Witteveen, Hester Hijn, (2009), "Explicit Mesh Deformation Using Inverse Distance Weighting Interpolation", *19th AIAA Computational Fluid Dynamics, San Antonio, Texas*.
- Xiangmin Jiao, (2007), "Face offsetting: A unified approach for explicit moving interfaces", *J. Computational Physics*, **220**, 612-625
- Xiangmin Jiao, Michael T. Heath, (2004), "Common-refinement-based data transfer between non-matching meshes in Multiphysics simulations", *Int. J. for Numerical Methods in Engineering*, **61**, 2402-2427
- 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