Computational analysis of flow field characteristics of a liquid rocket unshrouded impeller

H Negishi¹, S Ohno², T Fukuda³ and Y Ogawa⁴

¹Research Unit III, Research and Development Directorate, Japan Aerospace Exploration Agency (JAXA), Tsukuba, Ibaraki, 305-8505, Japan
²Ryoyu Systems Co. Ltd., Minato-ku, Tokyo, 108-0074, Japan
³Daiichi System Engineering Co. Ltd., Sakae Naka-ku, Nagoya, Aichi 460-0008, Japan
⁴H3 Project, Space Technology Directorate I, Japan Aerospace Exploration Agency (JAXA), Tsukuba, Ibaraki, 305-8505, Japan

Abstract. The unshrouded impeller applicable to liquid rocket pumps is a new technology compared to the conventional shrouded impeller widely used in existing liquid rocket engines. Although many past studies have investigated head/suction performance, the rotordynamic and axial forces acting on unshrouded impellers, the detailed flow field in unshrouded impellers, and particularly the influence of the compressibility of hydrogen have yet to be discussed. This paper describes the application of three-dimensional compressible Reynolds-averaged Navier-Stokes simulations to investigate the detailed flow field in a liquid rocket hydrogen pump with an unshrouded impeller. First, the computed results were validated relative to the experimental results obtained in the LE-X hydrogen pump test. Then, the detailed flow field was investigated by comparing the computed results at two different flow coefficients. The computed results agreed well with the experimental results, and revealed several key features of the flow field in the unshrouded impeller.

1. Introduction

In Japan, a new liquid rocket engine (called “LE-9”) has been developed as a booster engine for Japan’s next-generation launch vehicle (called “H3”) toward the maiden flight planned in 2020 [1]. The LE-9 engine employs an expander bleed cycle system using LH2/LOX propellant, which for many years has also been applied to the current upper-stage engine (called “LE-5B”) of the H-IIA/B launch vehicles. The reference operating condition of the LE-9 engine is characterized by a thrust of 1471 kN, combustion chamber pressure of 10 MPa, and a specific impulse of 426 s at sea level. Those characteristics are the most challenging targets in the world for expander-type cycle engines whose turbines are driven by heated fuel [2].

In LE-9 engine development, a high-performance and highly reliable fuel turbopump is one of the key components to realize the engine system. The expander bleed cycle system applied to the LE-9 engine has an open engine cycle, which requires less pump head than closed engine cycles, such as the staged combustion cycle of the LE-7A engine [3]. In considering such a feature, a two-stage inducer and a single-stage impeller are employed in the LE-9 fuel turbopump. This configuration makes it possible to shorten the shaft length and operate at a nominal rotational speed of 40,000 rpm below the second critical speed. Moreover, an unshrouded impeller is adopted to reduce the manufacturing cost, ease manufacturing restrictions, and ensure margins of structural strength [4-6].

The unshrouded impeller applicable to liquid rocket pumps is a new technology compared to the conventional shrouded impeller widely used in existing liquid rocket engines. In the early 2000s, NASA and the Boeing-Rocketdyne division conducted an intensive feasibility study on high head unshrouded
impeller technology based on both experimental and numerical work [7-8]. Williams et al. [7-8] showed the sensitivity of head and suction performance relative to a variation in tip clearance on the front side of an unshrouded impeller by conducting water flow tests. Steady and incompressible Reynolds-averaged Navier-Stokes (RANS) simulations were also performed for the water flow tests. The computed results agreed well with the experimental data, whereas noticeable discrepancies were observed between the numerical and experimental results in the case having a large tip clearance. Williams et al. [7] also conducted CFD parametric studies on such primary design parameters as the number of blades and tip clearance, and showed that the use of unshrouded impellers offers the potential to reduce the number of stages in a hydrogen pump. Hah et al. [9] conducted a coupled fluid/structural analysis by using unsteady RANS (URANS) and FEM to investigate the effects of unsteady loading due to an unsteady flow field on an unshrouded impeller structure. Their incompressible URANS results well predicted the head coefficient and hydraulic efficiency over a wide range of mass flow rates (65%, 80%, 100% and 120% design mass flow rates). Chen et al. [10] investigated unshrouded impeller rotordynamic fluid reaction force and coefficients by using unsteady CFD simulations. Their results showed that unshrouded impellers create unfavorable rotordynamic coefficients compared to shrouded impellers, whereas the axial thrust balance is similar to that of a shrouded impeller, which requires a proper configuration of the balance piston flow routing and wear ring location to ensure axial thrust balance. Regarding other studies related to unshrouded impellers, Nagao et al. [11] investigated rotordynamic force in a whirling motion by using active magnetic bearings. Suwa et al. [12] investigated the dynamic response of a fluid force acting on an axially oscillating unshrouded impeller front by using incompressible URANS.

As described above, many past studies have investigated head/suction performance and the rotordynamic and axial forces acting on unshrouded impellers. However, details of the flow field in the unshrouded impeller for the liquid rocket engine pump and its effect on rotordynamic force and axial thrust have yet to be discussed. As for axial thrust, Shimura et al. [13] showed that the compressibility of LH2 can cause large amplitude axial vibration in a shrouded impeller based on one-dimensional analysis of a balance piston. However, the influence of the compressibility of LH2 on the flow field and axial thrust in unshrouded impellers has not been discussed.

In this context, a compressible RANS simulation methodology that takes into account the real fluid properties of parahydrogen under wide-ranging pressure and temperature conditions was developed and applied to a liquid hydrogen pump with an unshrouded impeller of the LE-X engine [14-15], which was a precursor of the LE-9 engine, in our past study [16]. In the past study, the compressible unsteady RANS simulation revealed a relatively high Mach number region leading to a temperature rise in the outer radial region of the impeller and significant tip leakage flow structures emerging from the leading edge of impeller blades, which result in total pressure loss, at 98% design flow coefficient condition. In the current study, the compressible and steady RANS simulations were performed at two different flow coefficient conditions (100% and 83% design values) in order to investigate the influence of the flow coefficient on the detailed flow field in the unshrouded impeller. The computed results were validated relative to the experimental results obtained in the LE-X hydrogen pump test. Then, a comparison was made of the detailed flow fields at the different flow coefficients.

2. LE-X liquid hydrogen turbopump
The LE-X rocket engine [14-15] is a LOX/LH2 expander bleed-cycle engine designed and studied to demonstrate the technology of the LE-9 engine. The engine configuration and components of the LE-X are almost the same as those of the LE-9. Figure 1 shows the CAD model of the liquid hydrogen pump. The liquid hydrogen pump of the LE-X engine consists of a two-stage inducer, a guide vane, a single-stage unshrouded
impeller, a vaned diffuser, and a volute at the end [4-5]. As for the turbine side, a two-stage supersonic turbine was applied. The nominal rotational speed of the LE-X liquid hydrogen pump is 40,144 rpm, which is very similar to that of the LE-7A liquid hydrogen pump [3] and below the second critical speed. The nominal mass flow rate of LH2, rise in pressure, and shaft power are 49.7 kg/s, 17.6 MPa, and 16148 kW, respectively.

The self-balancing type axial thrust balancing system with the balance piston is applied. The balance piston consists of two orifices and a chamber surrounded by those two orifices, the impeller and rear-side casing, and is located on the rear side of the impeller. Several swirl brakes are equipped on the rear side of the impeller casing to adjust pressure distribution in the balance piston. And several balancing holes that connect the front and rear sides of the impeller are also processed on the impeller.

3. Numerical approach

3.1. Governing equations and basic numerical approach

Numerical simulations in this study were performed using the preconditioning density-based solver known as CRUNCH CFD [17-19], which was developed by CRAFT Tech [20] and extensively validated relative to rocket applications by JAXA [21-28]. CRUNCH CFD is an unstructured, multi-element flow solver based on the cell-vertex method, which permits a combination of hexahedral, tetrahedral, prismatic, and pyramidal elements in a mesh construction for viscous, real gas systems and multiphase gas-liquid systems. The governing equations are three-dimensional compressible Navier-Stokes equations. The Reynolds-averaged Navier-Stokes (RANS) approach was used to simulate liquid hydrogen flow inside the liquid hydrogen pump. In order to simulate rotating machinery, calculations of such rotating parts as the inducer and impeller were performed on a rotating reference frame, in which centrifugal and Coriolis forces are added to the momentum equations. The mixing plane model was used on interfaces between a moving part and a stationary part. The convection terms were discretized by the second order upwind scheme. The standard \( k-\varepsilon \) model with a wall function was applied as a turbulence model [19].

State variables such as density and transport properties such as viscosity were calculated as functions of both pressure and temperature. In the current study, the table lookup framework based on standard reference database 23 for pure fluids [29], which is available from the National Institute of Standards and Technology (NIST), was employed to calculate the thermodynamic and transport properties of parahydrogen.

3.2. Computational grid and boundary conditions

The computational domain consists of all the pump components described in Section II, including the leakage flows between the unshrouded impeller and the casings, as shown in figure 2. The number of blade passages and the geometry in the entire pump region were modeled to precisely reflect the actual pump configuration. An inlet pipe and a discharge pipe were placed upstream of the inducer and downstream of the volute, respectively, as shown in figure 3. There are approximately 530,000 computational grids for the inlet pipe, 2,370,000 for the inducer, 1,090,000 for the guide vane, 33,050,000 for the unshrouded impeller, 3,820,000 for the vaned diffuser, and 940,000 for the volute with the discharge pipe. Thus, there is a total of about 41.8 million computational grids. Each computational domain is linked with its upstream and downstream domains based on the non-conformal grid system.

The boundary conditions were specified based on the experimental results obtained in the LE-X hydrogen turbopump test as shown in figure 3. Table 1 shows the computational conditions considered in this study. Here, \( \phi \) is the flow coefficient, and \( \phi_d \) is its design value. The flow coefficient \( \phi \) is defined as \( Q/(ND_{tip}^3) \), where \( Q \) and \( D_{tip} \) denote the volume flow rate and the tip diameter of the impeller, respectively. In order to investigate the influence of the flow coefficient, two different flow coefficient conditions were taken into account. At 5D upstream of the inducer interface, static temperature and mass flow rate were given by considering a uniform velocity distribution. Here, \( D \) is the diameter of the inlet
duct. It should be noted that a pre-rotation and the incoming boundary layer were not taken into account in the present calculations. Static pressure was specified at the downstream boundary of the discharge pipe and at the exit of the leakage passage on the rear side of the impeller. A no-slip and adiabatic boundary condition was prescribed on the solid walls in the entire computational domain.

![Diagram of the LE-X liquid hydrogen pump and computational grids.](image)

**Figure 2.** Schematic of the LE-X liquid hydrogen pump and the computational grids.

| Symbol               | Unit | Case 1 | Case 2 |
|----------------------|------|--------|--------|
| Flow coefficient ratio | \( \phi/\phi_d \) | -      | 1.00   | 0.83   |
| Number of rotation   | \( N \) | rpm    | 31159  | 21224  |
| Inlet temperature    | TIF  | K      | 20.5   | 20.5   |
| Mass flow rate       | WF   | kg/s   | 39.4   | 22.2   |

**Figure 3.** Entire computational domain and boundary conditions.

**Table 1.** Computational conditions.
4. Results and discussion

4.1. Comparison with the LE-X liquid hydrogen turbopump experiment

Figure 4 shows a comparison of normalized static pressures at multiple measurement points around the unshrouded impeller for the two different flow coefficient cases. The computed normalized static pressure contours are also displayed together on the meridional plane. In the figure, \( p \) is the static pressure, \( \rho \) is density at the impeller inlet, and \( U_{tip} \) is tip velocity of the impeller. Regardless of the experimental and computational results, the normalized static pressures around the impeller in the low flow coefficient case (case 2) are higher than those in the design flow coefficient case (case 1). As for a comparison between the computational and experimental results, the computed pressures in the leakage flow passage as well as the main flow passage agreed well overall with the experimental results. The pressures were well predicted within a 5% difference in case 1 (\( \phi/\phi_d = 1.0 \)), and within 3% in case 2 (\( \phi/\phi_d = 0.83 \)).

With regard to static temperature prediction, the computed outlet temperature of the pump was predicted as 28.6 K in case 1 and 24.5 K in case 2, which were in reasonable agreement with the experimental results within a 3-K difference. After all, it was confirmed that the computed results can predict static pressures and temperatures around the unshrouded impeller very well, and can be used for a detailed discussion of the flow fields in the following section.

![Figure 4. Comparison of normalized static pressures at multiple measurement points inside the pump. (The computed static pressure contour is also displayed on the meridional plane.)](image-url)

4.2. Flow field characteristics

This section discusses the fundamental flow field characteristics in the unshrouded impeller by comparing the computed results of the two different flow coefficient cases.

In figure 5, (a), (b), (c), and (d) show the normalized static pressure, static temperature, Mach number, and density contour plots on the mid-plane of the impeller and diffuser, respectively. In figure 5 (a), almost the same normalized static pressure distributions are obtained qualitatively regardless of the flow coefficient. As described in figure 4, the normalized static pressure in the entire region at the low flow coefficient (\( \phi/\phi_d = 0.83 \)) is higher than that at the design flow coefficient (\( \phi/\phi_d = 1.0 \)). It should be noted that in reality the static pressure in case of \( \phi/\phi_d = 0.83 \) is lower than that in case of \( \phi/\phi_d = 1.0 \).

In the outer radial region of the impeller, a rise in static temperature can be clearly seen in both cases in figure 5 (b), and reaches about 8 K in the case of \( \phi/\phi_d = 1.0 \), compared to about 4 K in the case of \( \phi/\phi_d = 0.83 \). The regions of those temperature rises in the impeller correspond to the region of high Mach number in the impeller region as shown in figure 5 (c). In the case of \( \phi/\phi_d = 1.0 \), the Mach number reached up to 0.36, which means the compressibility of the flow can weakly influence the flow field characteristics, whereas the Mach number is less than 0.3 in the case of \( \phi/\phi_d = 0.83 \), which means that the compressibility can be neglected. A further rise in temperature is observed in the diffuser region in both cases. The rise in temperature can be attributed to the work done by pressure due to the compressibility of liquid hydrogen and the rise in entropy due to friction and mixing/diffusion losses.
inside the impeller and diffuser regions. In the case of $\phi/\phi_d = 1.0$, a rise in temperature can be induced by both effects due to a relatively high Mach number, whereas a rise in temperature can only be caused by the rise in entropy in the case of $\phi/\phi_d = 0.83$ due to a low Mach number.

In figure 5 (d), high density regions are observed around the pressure side of the impeller blades and

![Figure 5. Computed static pressure, static temperature, Mach number, and density contours on the mid-plane.](image-url)
the diffuser vane region in both flow coefficient cases. The important point here is that overall local density increased in the radial direction from the impeller toward the outlet volute, even though the local static temperature also increased in the same direction as shown in figure 5 (b). In general, density decreases as static temperature rises under the constant static pressure condition, whereas density increases as static pressure rises under the constant static temperature condition. Therefore, the computed results shown in figure 5 (d) indicate that the increase in density is attributed to the significant rise in static pressure in the pump. The increase in density reached about 5 kg/m$^3$ in the case of $\phi/\phi_d = 1.0$, and about 2.5 kg/m$^3$ in the case of $\phi/\phi_d = 0.83$ due to the difference in the increase of static pressure.

Figure 6 shows the computed relative velocity magnitude distribution and its close-up view on the mid-plane. In both cases, low velocity regions are observed downstream of the suction side of the full and splitter blades. The unsteady RANS simulation performed in our past study [16] showed that the sizes of the low velocity regions vary over time and are significantly affected by the impeller-diffuser interaction. A comparison between the results of $\phi/\phi_d = 0.83$ and 1.0 shows that the low velocity regions are larger in size in the case of $\phi/\phi_d = 0.83$ than those in the case of $\phi/\phi_d = 1.0$.

Figure 7 shows a comparison of streamlines and surface restricted lines in the unshrouded impeller with vortex cores detected by eigenmode analysis [30]. The pink and blue streamlines represent the tip leakage flow emerging from the tip clearance on the full and splitter blades, respectively. The yellow streamlines correspond to the leakage flow passing through the balance hole from the rear part of the impeller. The black lines denote the surface restricted lines. Vortex cores are colored by the normalized helicity ($H_n$) defined as follows:

$$H_n = \frac{\widehat{\xi} \cdot \widehat{w}}{|\widehat{\xi}||\widehat{w}|}$$

(1)

where $\widehat{\xi}$ and $\widehat{w}$ denote the vectors of absolute vorticity and relative flow velocity, respectively. Normalized helicity $H_n$ is the cosine of the angle between the absolute vorticity and relative velocity vectors. That means that the magnitude of normalized helicity $H_n$ takes the value of unity anywhere the streamwise vortex exists, and its sign indicates the swirl direction of the vortex relative to the streamwise velocity component based on the right-hand rule. Thus, normalized helicity $H_n$ can evaluate the nature of a vortex quantitatively.

In the computed results for both cases, almost the same flow structures can be seen. Vortex cores of almost +1 in the normalized helicity are clearly observed near the tips of the full and splitter blades, respectively. Those vortex cores are wrapped by the tip leakage streamlines. Those results indicate the
presence of a streamwise vortex, representing the tip leakage vortex rolling up from the leading edges of the impeller full and splitter blades. At the middle part of the impeller passage along the full blade, the tip leakage vortex core disappears and then the secondary leakage vortex core appears downstream of the passage.

The surface restricted lines in figure 7 indicate secondary flows toward the hub along the suction side of the full and splitter blades in both cases. This flow is caused by the presence of the tip leakage vortices, which deflect the main flow toward the hub along the full blade. Conversely, another secondary flow toward the casing is observed along the pressure side of the full and splitter blades.

Figure 8 compares the distributions of total pressure loss coefficient $\zeta_p$ on several constant radius planes in the impeller passages. The total pressure loss coefficient $\zeta_p$ is defined as:

$$\zeta_p = \frac{\omega(r_{c_{\text{in}}} - r_{c_{\text{in},m}}) - (P - P_{\text{in}})/\rho}{U_{\text{tip}}/2}$$  \hspace{1cm} (2)$$

where $\omega$ denotes the angular velocity, $r$ the radius from the axis of rotation, $c_{\theta}$ the absolute tangential velocity component, $P$ the total pressure, $\rho$ the density, $U_{\text{tip}}$ the blade tip velocity, and the subscript $\text{in}$ denotes the impeller inlet. Based on equation (2), higher $\zeta_p$ means higher total pressure loss. Figure 8

![Figure 6. Relative velocity magnitude distribution on the mid-plane in the impeller region.](image-url)

(The entire impeller region is shown on the left; a close-up view of the region is shown on the right.)
shows that in both cases high loss fluid accumulates around the tip leakage vortex regions along the casing and suction sides of the full and splitter blades. The high loss region gradually grows larger in the streamwise direction due to the complex secondary flow behavior downstream of the impeller passages as shown in figure 7. Those results indicate that the total pressure loss in the unshrouded impeller is caused by the tip leakage vortices and subsequent secondary flows along the passage. The total pressure loss leads to a rise in entropy, which results in a rise in temperature as shown in figure 5 (b). Overall, the total pressure loss coefficient at the impeller outlet in the case of \( \phi / \phi_d = 1.0 \) is higher than that in the case of \( \phi / \phi_d = 0.83 \), which is attributed to the stronger secondary flow structures in the case of \( \phi / \phi_d = 1.0 \) as shown in figure 7 (a).

Figure 7. Vortex cores colored by normalized helicity with streamlines. (Streamlines and surface restricted lines are also shown, where the pink and blue lines denote tip leakage flows emerging from the leading edges of the full and splitter blades, respectively. The yellow lines represent the streamlines of the leakage flow through the balance hole. The black lines denote the surface restricted lines.)

Figure 8. Comparison of total pressure loss coefficient distributions on several constant radius planes.
5. Conclusions
Three-dimensional compressible Reynolds-averaged Navier-Stokes simulations of the LE-X liquid hydrogen pump were performed in order to understand the detailed flow field in an unshrouded impeller. In particular, the influence of the flow coefficient on the flow field was discussed based on the computed results.

First, the computed results were validated relative to the experimental results obtained in the LE-X liquid hydrogen turbopump test. The pressures in the main and leakage flow passages around the unshrouded impeller were well predicted within a 5% difference in the design flow coefficient case, and within 3% in the 83% design flow coefficient case. As for temperature, the computed outlet temperature of the pump agreed well with the experimental results within a 3-K difference for both flow coefficient cases.

A detailed investigation of the flow field revealed several key features of the flow field inside the unshrouded impeller for the two different flow coefficient cases. Regardless of the flow coefficient being considered, the computed results predicted a rise in temperature and a variation in density due to the work done by pressure and friction, as well as mixing/diffusion losses on the impeller front side. In the design flow coefficient case, the maximum Mach number reached up to 0.36, which means the compressibility of hydrogen weakly influences the flow field, and the rise in static temperature and density was higher than that in the 83% design flow coefficient case. Conversely, the Mach number is lower than 0.3 in the 83% design flow coefficient case, which means that the compressibility can be neglected. Furthermore, the computed results clarified that the flow field on the impeller front side is significantly influenced by the tip leakage vortices rolling up from the leading edges of the full and splitter blades. Although almost the same flow fields were observed regardless of the flow coefficient, the total pressure loss coefficient at the impeller outlet at the design flow coefficient was higher than that at the 83% design flow coefficient, which is mainly due to strong secondary flow structures that appear downstream of the impeller blades.

Acknowledgements
This study was supported by Akiko Kotani of Matesis Co., Ltd., and Dr. Ashvin Hosangadi of Combustion Research and Flow Technology (CRAFT Tech). All simulations in this study were performed on the JAXA Supercomputer System Generation 2 (JSS2). The authors would like to express their sincere appreciation for the substantial support provided by these individuals and JSS2.

References
[1] Tamura A, Niitsu M, Kamiya T, Sato A and Yoshikawa K 2014 Mitsubishi Heavy Industries Technical Review 51 No. 4 pp 20-25
[2] Watanabe D, Manako H, Onga T, Tamura T, Ikeda K and Isono M 2016 Mitsubishi Heavy Industries Technical Review 53 No. 4 pp 28-35
[3] Matsuyama K, Ito T, Ohigashi H, Yasui M and Manako H 2002 Mitsubishi Heavy Industries Technical Review 39 No. 2 pp 51-56
[4] Mizuno T, Kobayashi S and Oguchi H 2010 IHI Engineering Review 43 No. 1 pp 43-46
[5] Kojima M, Kurosu A, Uchiumi M, Takada S, Noda K and Mizuno T 2011 47th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit (San Diego) AIAA Paper 2011-5932
[6] Azuma N, Ogawa Y, Aoki K, Kobayashi T, Okita K, Mizuno T, Niiyama K and Shimiya N 2017 AIAA Propulsion and Energy (Atlanta) AIAA Paper 2017-4927
[7] Williams R W, Skelley S E, Stewart E T, Droge A R, Prueger G H, Chen W-C and Williams M 2000 36th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit (Huntsville) AIAA Paper 2000-3243
[8] Williams R W, Skelley S E, Chen W-C and Williams M 2001 37th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit (Salt Lake City) AIAA Paper 2001-3398
[9] Hah C, Loellbach J, Owen A and Khandelwal S 2001 37th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit (Salt Lake City) AIAA Paper 2001-3477
[10] Chen W-C, Williams M, Paris J K, Prueger G H and Williams R 2001 37th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit (Salt Lake City) AIAA Paper 2001-3399

[11] Nagao N, Eguchi M, Uchiumi M and Yoshida Y 2011 4th European Conference for Aerospace Sciences (Saint Petersburg)

[12] Suwa N, Kimura T, Kawasaki S, Goto K and Uchiumi M 2017 31st International Symposium on Space Technology and Science (Matsuyama)

[13] Shimura T, Kawasaki S, Uchiumi M, Kimura T, Hayashi M and Matsui J, ASME J. Fluids Engineering 135 011105

[14] Kurosu, A, Yamanishi N, Sunakawa H, Nishimoto M, Okita K, Kumakawa A, Ogawara A, Onga T and Manako H 2008 44th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit (Hartford) AIAA Paper 2008-4665

[15] Atsumi M, Yoshikawa K, Ogawara A and Onga T 2011 Mitsubishi Heavy Industries Technical Review 48 No. 4

[16] Negishi H, Ohno S, Ogawa Y, Aoki K, Kobayashi T, Okita K and Mizuno T 2017 53rd AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit (Atlanta) AIAA Paper 2017-4930

[17] Hosangadi A, Lee R A, York B J, Sinha N and Dash S M 1996 J. of Propulsion and Power 12 No. 3 pp 494-503

[18] Vineet A, Shipman J D, Arunajatesan S and Hosangadi A 2003 J. of Propulsion and Power 19 No. 5 pp 945-952

[19] Hosangadi A, Ahuja V, Ungewitter R J and Busby J 2007 J. of Propulsion and Power 23 No. 6 pp 1225-1234

[20] URL: http://www.craft-tech.com/index.html

[21] Daimon Y, Ohnishi Y, Negishi H and Yamanishi N 2009 45th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit (Denver) AIAA Paper 2009-5492

[22] Daimon Y, Negishi H and Yamanishi N 2010 46th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit (Nashville) AIAA Paper 2010-6723

[23] Daimon Y, Negishi H, Yamanishi N, Nunome Y, Sasaki M and Tomita T 2011 47th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit (San Diego) AIAA Paper 2011-5625

[24] Negishi H, Daimon Y, Kawashima H and Yamanishi N 2012 48th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit (Atlanta) AIAA Paper 2012-4122

[25] Negishi H, Daimon Y, Kawashima H and Yamanishi N 2013 49th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit (San Jose) AIAA Paper 2013-3997

[26] Negishi H, Daimon Y, Negoro N and Kurosu A 51th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit (Orlando) 2015 AIAA Paper 2015-3760

[27] Tani N, Yamanishi N and Tsujimoto Y 2012 J. of Fluids Engineering 134 Issue 2

[28] Kawatsu K, Tani N and Yamanishi N 2013 51th AIAA Aerospace Science Meeting including the New Horizons Forum and Aerospace Exposition (Dallas/Ft. Worth Region) AIAA Paper 2013-0989

[29] NIST Standard Reference Database 23 NIST Reference Fluid Thermodynamic and Transport Properties Database (REFPROP): Version. 8.0 URL: https://www.nist.gov/srd/refprop

[30] Haimes R and Kenwright D 1999 14th AIAA Computational Fluid Dynamics Conference (Norfolk) AIAA Paper 99-3288