Numerical study on cavitating flow-induced vibration of liquid rocket engine inducer

Q Wu¹², B Huang², G Wang², S Cao¹, H Zhang²
¹ Department of Energy and Power Engineering, Tsinghua University, Beijing, 100084, China
² School of Mechanical Engineering, Beijing Insititute of Technology, Beijing, 100081, China

Email: wuqin919@163.com

Abstract. The cavitating flow-induced vibrations have become one of the major issues for the operation safety and stability of the liquid rocket engine. The objective of this paper is to numerically investigate the cavitating flow around a four-blade inducer, with focus on the cavitating flow-induced vibration characteristics. In the numerical simulation, the curvature correction turbulence model and the Zwart cavitation model is used for the simulation of the flow field. The loose coupled method is adopted for the prediction of the fluid structure interaction, including the calculation of the fluid forces based on the multiphase fluid dynamics, computation of the structural deformation via the governing equation of the structural motion, and then the update the fluid and structure mesh. The results showed that good agreement has been obtained between the experimental and numerical results. The reverse flow vortex cavitation develops and rotates with the blade, but with a much lower rotational speed than that of the blade. The vibration of the hydrocone is mainly affected by the blade rotation of the inducer, and the blade vibration is mainly affected by the reverse flow vortex cavitation.

1. Introduction
In recent years, the demands for the thrust vehicle increases with the needs of the exploration and development for outer space. In order to achieve high thrust vehicle, it needs to increase further the performance of rocket engines. A turbopump is employed to supply the low temperature propellants to the combustion chamber operated under extremely high pressure. And an inducer is attached to the turbopump to increase the efficiency. Design of any space vehicle component is always guided by minimum size, and consequently, the size constraint on the turbopump solicits high impeller speed. Such high speed likely results in cavitation around the inducer blades [1-3]. As the cavitation will lead to many problems such as pressure pulsation, vibration, noise and erosion, it has become a primary concern for the hydraulic machine and many other fields [4-6]. Especially in the axial inducers used for liquid-propellant rocket turbopumps, typically working under cavitating conditions, the flow instabilities and flow-induced vibration can seriously degrade the performance of the machine or even cause its rapid failure. Hence, the cavitating flow-induced vibration problem has become the decisive factor affecting the operation safety and stability of the liquid-propellant rocket engine.
The objective of this paper is to numerically investigate the cavitating flow around a four-blade inducer, with focus on the cavitating flow-induced vibration characteristics.

2. Numerical Method

2.1. Governing equations

The incompressible and unsteady Reynolds Average Navier-Stokes (URANS) equations are used due to its balance between the accuracy and the computational cost.

\[
\frac{\partial (\rho_m)}{\partial t} + \frac{\partial (\rho_m u_j)}{\partial x_j} = 0
\]

\[
\frac{\partial (\rho_m u_j)}{\partial t} + \frac{\partial (\rho_m u_j u_i)}{\partial x_i} = -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_i} \left[ \left( \mu_m + \mu \right) \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right]
\]

\[
\rho_m = \rho_l \alpha_l + \rho_v \alpha_v
\]

\[
\mu_m = \mu_l \alpha_l + \mu_v \alpha_v
\]

where \(\rho_m\) is the mixture fluid density, \(\rho_l\) and \(\rho_v\) are the liquid and vapor densities respectively, \(u\) is the velocity, \(p\) is the pressure, \(\mu_m\) is the mixture laminar viscosity, \(\mu_l\) and \(\mu_v\) are respectively the liquid and vapor dynamic viscosity, \(\mu_t\) is the turbulent viscosity. The subscripts \(i, j\) denote the directions of the Cartesian coordinates.

2.2. Curvature correction turbulence model

The original \(k-\epsilon\) turbulence model proposed by Launder and Spalding [7] shows two partial differential equations for the transport of the turbulence kinetic energy \(k\) and dissipation rate \(\epsilon\):

\[
\frac{\partial (\rho_m k)}{\partial t} + \frac{\partial (\rho_m u_j k)}{\partial x_j} = \rho_l \frac{\partial \epsilon}{\partial x_j} + \rho_l \frac{\partial \left( \mu_l \frac{\partial k}{\partial x_j} \right)}{\partial x_j} - \rho_m \frac{\partial \epsilon}{\partial x_j} - C_{e1} \rho_m \frac{\partial \epsilon^3}{\partial x_j} + \rho_m \frac{\partial \left( \mu_t \frac{\partial \epsilon}{\partial x_j} \right)}{\partial x_j}
\]

\[
\frac{\partial (\rho_m \epsilon)}{\partial t} + \frac{\partial (\rho_m u_j \epsilon)}{\partial x_j} = C_{e1} \frac{\epsilon}{k} - C_{e2} \rho_m \frac{\epsilon^2}{k} + \frac{\partial}{\partial x_j} \left[ \mu_t \frac{\partial \epsilon}{\partial x_j} \right]
\]

The eddy viscosity is defined as

\[
\mu_t = \frac{C_{\mu} \rho_m k^2}{\epsilon}, \quad C_{\mu} = 0.09
\]

As suggested by Shur[8], the correction function is defined to describe the streamline curvature effect in strong vertical flows:

\[
f_r = \left(1 + C_{r1} \right) \frac{2r^*}{1 + r^*} \left[ 1 - C_{r3} \arctan(C_{r2} \tilde{r}) \right] - C_{r1}
\]

Then the production term of turbulent kinetic energy \(P_k\) can be revised as:

\[
P_k \rightarrow P_k \cdot f_r
\]

2.3. Cavitation model

The cavitation process is governed by the mass transfer equation for the conservation of liquid/vapor volume/mass fraction, which can be defined as:

\[
\frac{\partial (\rho_l \alpha_l)}{\partial t} + \frac{\partial (\rho_l \alpha_l u_j)}{\partial x_j} = \dot{m}^+ + \dot{m}^-
\]

In this work, the Zwart cavitation model [9] is used, which is derived from Rayleigh-Plesset equation and provides the rate equation controlling vapor generation and condensation. The representative liquid-vapor evaporation and condensation rates for this category can be shown as:
\[ \dot{m}^- = -C_{dest} \frac{3\alpha_{vuc} (1-\alpha_v) \rho_v}{R_h} \left( \frac{2}{3} \frac{p_v - p}{\rho_l} \right)^{\frac{1}{2}}, p < p_v \] (11)

\[ \dot{m}^+ = C_{prod} \frac{3\alpha \rho_l}{R_h} \left( \frac{2}{3} \frac{p - p_v}{\rho_l} \right)^{\frac{1}{2}}, p > p_v \] (12)

where \( C_{dest} \) and \( C_{prod} \) are the constant generation and re-conversion rate of vapor. All the model constants are assumed as the default values in CFX (Zwart et al., 2004).

2.4. Fluid structure interaction model

The governing equation of the structural system motion can be written as:

\[ M\ddot{q}(t) + C\dot{q}(t) + Kq(t) = Q(t) \] (13)

Where \( \ddot{q} \), \( \dot{q} \) and \( q \) are the acceleration, velocity and displacement, \( M \) is the mass matrix, \( C \) is the damping matrix, \( K \) is the stiffness matrix, \( Q(t) \) is the external excitation force acting on the structural system.

The loose coupled method is adopted for the prediction of the fluid structure interaction, including the calculation of the fluid forces based on the multiphase fluid dynamics, computation of the structural deformation via the governing equation of the structural motion, and then the update the fluid and structure mesh.

2.5. Numerical setup

The four-bladed, tapered-hub, variable-pitch, high-head inducer, DAPAMITO4[10], has been studied in this work, with the design speed of 3000r/min and the design flow coefficient of 0.07. The computational domain includes three parts: the entrance section, the inducer section and the exit section, as shown in Figure 1. Figure 2 shows the corresponding mesh distribution, with the nodes numbers are 340 thousand, 1 million and 200 thousand for the inlet, the inducer and the exit sections.

Figure 1. Computational domain.

(a) Inlet section  (b) Outlet section  
(c) Inducer section

Figure 2. Mesh distributions.
3. Results and discussions

3.1. Numerical validation

To validate the numerical method used in this work, Figure 3 shows the comparison of experimental and numerical results at noncavitating and cavitating flow. Figure 3(a) compares the experimental measured and numerical predicted head coefficients of the inducer DAPAMITO4 with the speed of 1750r/min and different flow coefficient at noncavitating flow. Figure 3(b) compares the experimental and numerical head coefficients with the design speed of 3000r/min and the flow coefficient of 0.053 at cavitating flow. Reasonable agreement can be obtained between the experimental and numerical results, especially for the conditions with large flow coefficients or large cavitation numbers. With the decrease of the cavitation number, the head coefficient of the inducer reduces and an intermediate head drop occurs under the condition with $\sigma=0.077$.

Figure 3. Comparison of experimental and numerical results at noncavitating and cavitating flow.

Figure 4 presents the experimental observed and numerical predicted cavitation patterns with different cavitation numbers. Good agreement has been obtained between the experimental and numerical results. When the cavitation number $\sigma=0.288$, the tip vortex cavitation occurs at the leading edge of the blade. With the decrease of the cavitation number, the leading edge vortex cavitation develops. For the conditions with the cavitation number $\sigma=0.092$ and $\sigma=0.060$, the cloud cavitation can be observed, which is corresponding to the significant decrease of the head coefficient.
3.2. Cavitation flow-induced vibrations

Figure 5 shows the evolution of the cavity patterns around the inducer. It can be observed that the cavity area attached on the blade surface and the reverse flow vortex cavitation vary with the time. When \( t=t_0 \), the reverse flow vortex cavitation begins to form and distributes asymmetrically around the blades. From \( t=t_0 \) to \( t=t_0+6/10T \), the reverse flow vortex cavitation develops and rotates with the blade, but with a much lower rotational speed than that of the blade. When \( t=t_0+7/10T \), a new small-scale cavity begins to form in front of the developed reverse flow vortex cavity, and with the development of the new cavity, the previous reverse flow vortex cavity shrinks to some extent. During the rotational cycle of the inducer, the reverse flow vortex cavitation location rotates for about 90°, with the rotation frequency of 12.5Hz. The frequency is similar to that observed by experiment[10], which validates further the feasibility of numerical method.

Figure 4. Cavitation patterns in the inducer at different cavitation numbers (experimental data are referred to [10]).
Figure 5. Evolution of the cavity patterns around the inducer.

To investigate the structural vibration characteristics, the vibration amplitude at the peak of the hydrocone has been monitored. Figure 6 shows the transient radial deformation of the hydrocone peak and corresponding power spectrum. The periodical variation of the vibration amplitude can be obviously observed. From the results in the frequency domain, the main vibration frequency of the hydrocone is 200Hz, which is accordance to the blade frequency of the inducer. While there is also an amplitude peak at $f=12.45$ Hz, which means that the vibration of the hydrocone mainly affected by the blade rotation of the inducer, and the reverse flow vortex cavitation has small effect on the vibration of the hydrocone.

Figure 6. Transient radial deformation of the hydrocone peak and corresponding power spectrum.

To further investigate the structural vibration characteristics, Figure 7 shows the evolution of the axial displacement of the blade. It can be found that the maximum displacement occurs at the leading edge of the blade tip, and the displacement reduces gradually from the leading edge of the blade tip to the blade root. Moreover, due to the pressure fluctuation in the flow field and the rotation of the blade, the maximum displacement position also varies with the time.
Figure 7. Evolution of the axial displacement of the blade.

Based on the Fast Fourier Transform of the vibration amplitude of the leading edge of the blade tip, as shown in Figure 8, it is different from the spectrum of the hydrocone peak radial deformation. The main vibration frequency for all the blades are about 12.5Hz, which is consistent with the reverse flow vortex cavitation frequency. The blade vibration is mainly affected by the reverse flow vortex cavitation.

Figure 8. Vibration amplitude spectrum of the leading edge of the blade tip.

4. Conclusions
In this work, the cavitating flow around a four-blade inducer, with focus on the cavitating flow-induced vibration characteristics has been numerically investigated. The main conclusions are including:

Good agreement has been obtained between the experimental and numerical results. With the decrease of the cavitation number, the leading edge vortex cavitation develops, corresponding to the reduction of the head coefficient.
The cavity area attached on the inducer blade surface and the reverse flow vortex cavitation vary with the time. The reverse flow vortex cavitation develops and rotates with the blade, but with a much lower rotational speed than that of the blade.

The vibration of the hydrocone is mainly affected by the blade rotation of the inducer, and the blade vibration is mainly affected by the reverse flow vortex cavitation.

Acknowledgement
This work is supported by National Postdoctoral Program for Innovative Talents (Grant No. BX201700126), the China Postdoctoral Science Foundation (No. 2017M620043), National Natural Science Foundation of China (Grant No. 51679005) and Beijing Municipal Natural Science Foundation of China (Grant No. 3172029).

References
[1] Utturkar Y, Wu J, Wang G, Shyy W. Recent progress in modeling of cryogenic cavitation for liquid rocket propulsion. Prog Aerosp Sci 2005, 41: 558-608.
[2] Coutier-Delgosha O, Morel P, Fortes-Patella R, et al. Numerical simulation of turbopump inducer cavitating behavior[J]. International Journal of Rotating Machinery, 2005, 2005(2): 135-142.
[3] Ugajin H, Watanabe O, Kawai M, Kobayashi S. Numerical simulation of cavitating flow in inducers. 40th Jt Propuls Conf Exibit 2004, 1-7.
[4] Wu Q, Huang B, Wang G, Cao S. The transient characteristics of cloud cavitating flow over a flexible hydrofoil. Int J Multiph Flow 2018, 99, 162-173.
[5] Arndt, R.E.A., Pennings, P., Bosschers, J., Van Terwisga, T., 2015. The singing vortex. Interface Focus 5, 1-11.
[6] Ausoni, P., Farhat, M., Escaler, X., Egusquiza, E., Avellan, F. Cavitation Influence on von Kármán Vortex Shedding and Induced Hydrofoil Vibrations. J. Fluids Eng., 2007, 129, 966-973.
[7] Launder BE, Spalding D, The numerical computation of turbulent flows, Comput. Methods Appl. Mech. Eng. 1974, 3(2), 269-289.
[8] Shur ML, Strellets MK, Travin AK, et al. Turbulence modeling in rotating and curved channels: assessing the spalart-shur correction. AIAA Journal, 2000, 38(5): 784-792.
[9] Zwart P, Gerber A, Belamri T. A two-phase flow model for predicting cavitation dynamics. Fifth Int. Conf. Multiph. Flow, Yokohama, Japan, 2004.
[10] Cervone A, Torre L, Pasini A, et al. Cavitation and flow instabilities in a 4-bladed axial inducer designed by means of a reduced order analytical model. 47th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit. 2011: 5781.