

Postprint: Prediction of Cavitation Characteristics and Pressure Pulsation for a High-Speed Turbopump at Rated Operating Condition

Authors: Zhang Haochen, ZUO Zhigang, Liu Shuhong, Wu Yulin

Date: 2017-11-07T00:00:00+00:00

Abstract

This study performs cavitation flow field calculations for a turbopump using the SST $k-\omega$ turbulence model and the ZGB cavitation model, obtains its hydraulic performance curves and cavitation performance curves, and analyzes the internal flow field and pressure fluctuation characteristics under design conditions. The results indicate that cavitation is prone to occur at three locations in the turbopump, the pressure fluctuations upstream and downstream of the inducer are primarily determined by the inducer blade passing frequency, and the pressure fluctuations inside the volute are significantly influenced by rotor-stator interaction.

Full Text

Preamble

Characteristics of Cavitation and Pressure Fluctuation at Rated Condition in High-speed Turbopump

Zhang Hao-Chen, Zuo Zhi-Gang, Liu Shu-Hong, Wu Yu-Lin
(State Key Laboratory of Hydroscience and Engineering, Tsinghua University, Beijing 100084, China)

Abstract: This paper used SST $k-\omega$ turbulence model and ZGB cavitation model to simulate cavitation flow field in a turbopump. Its hydraulic performance curves and cavitation performance curves were obtained and the inner flow distribution and pressure fluctuation characteristics in this turbopump were analyzed under the design point. Results show that there are three cavitation regions, and the pressure fluctuation in the front of inducer and behind the inducer is determined by the inducer blade passing frequency. Rotor-stator interaction impacts the pressure fluctuation in the volute.

Key words: turbopump; inducer; cavitation; pressure fluctuation; numerical simulation

For liquid rocket engines, the turbopump is a critical component. However, as turbopumps develop toward higher speeds and power densities, increasingly stringent requirements are placed on their cavitation performance. Inducers serve as the primary means to improve the cavitation performance of main pumps, yet stability issues can still arise even when operating under certain cavitation conditions. The failure of Japan's H-II rocket launch in 1999, whose engine wreckage was recovered from the ocean, was attributed to cavitation instability in the LE-7 engine's turbopump inducer, leading to engine malfunction and ultimately the loss of the launch vehicle [1]. Similarly, Pratt & Whitney's development of an improved high-pressure liquid oxygen turbopump for NASA's Space Shuttle Main Engine encountered severe vibration caused by cavitation, which damaged inducer blades and forced premature termination of testing [2]. Consequently, investigating the internal cavitation flow field and pressure fluctuations in turbopumps is essential for analyzing and optimizing flow stability.

Due to the difficulty of conducting turbopump cavitation experiments, numerical simulation has become the primary method for studying internal flow. M. M. Athavale et al. [3] utilized CFD-ACE+ software with a full cavitation model and standard k -turbulence model, where the vapor transport equation was derived from the Rayleigh-Plesset equation and other conditions, to compute the flow field in the SSME turbopump inducer and centrifugal pump. By capturing subtle flow field variations, they identified changes in cavity length and cavity locations under different conditions. Hosangadi A et al. [4] and Dorney D et al. [5] employed the standard k -model to perform numerical calculations on a single inducer, simulating internal cavitation and analyzing its instability mechanisms. This paper uses Fluent software to simulate the cavitation flow field in a turbopump, analyzing internal cavitation regions and their causes while discussing factors influencing internal pressure fluctuations.

1 Numerical Calculation Method

The turbopump was modeled and divided into five components: inlet section, inducer, centrifugal impeller, volute, and outlet section. The full flow path geometry model is shown in Figure 1 [Figure 1: see original paper]. Hexahedral meshes were used for the inlet, outlet, and volute sections, while tetrahedral meshes were employed for the inducer and centrifugal impeller, with refinement applied to the gap between the inducer blade tip and the wall. The main turbopump parameters and computational boundary conditions are listed in Table 1. To ensure result accuracy, grid independence verification was performed, as shown in Figure 2 [Figure 2: see original paper], and calculations were conducted using 5.47 million cells.

2 Steady-State Calculation Results Analysis

Steady-state numerical simulation was employed to predict the turbopump's external performance curves, as shown in Figure 3 [Figure 3: see original paper]. For this turbopump, maximum efficiency occurs near 1.1Q₀, with an efficiency of 67% at the design condition and a head of 532 m. By varying the inlet pressure, the turbopump's cavitation performance curves were obtained (Figure 4 [Figure 4: see original paper]). The critical cavitation point is generally defined as the operating point where efficiency drops by 3%. The results indicate that the design condition corresponds to the critical cavitation point, suggesting that the turbopump studied still has potential for optimization.

The pressure distribution on the suction and pressure surfaces of two inducer blades is shown in Figure 5 [Figure 5: see original paper]. Two low-pressure regions exist within the inducer: one at the blade inlet near the tip and another at the blade outlet near the hub. The pressure variation trends are consistent across both blades for both suction and pressure surfaces, with the pressure surface exhibiting higher pressure than the suction surface.

Figure 6 [Figure 6: see original paper] illustrates the streamline distribution at the inducer mid-span plane, revealing the presence of a backflow vortex at the inducer blade inlet—this is the inducer inlet backflow phenomenon. Inducer inlet backflow comprises two types: vortices perpendicular to the axial plane and swirling flow around streamlines. This occurs because the rotating inducer blades exert non-uniform forces on the fluid. Centrifugal effects create a pressure differential between the blade tip and hub, but because the blades continuously rotate, this dynamic pressure differential persists, causing fluid to flow back from the blade inlet tip to the inlet pipe section. This backflow induces rotational motion in the main flow. Additionally, under actual operating conditions, the real flow angle is smaller than the design flow angle, creating a non-uniform circumferential velocity component at the inducer blade leading edge that generates streamline-oriented vortices. Inducer inlet backflow leads to localized low pressure.

High-speed turbopumps have small flow coefficients and are prone to stall. Stall phenomena first occur at the hub location, causing uneven pressure and velocity distributions at the outlet, as shown in Figure 8 [Figure 8: see original paper]. Due to centrifugal effects in the inducer, liquid is thrown to the tip region, resulting in higher pressure at the tip and lower pressure at the hub. This creates a vortex region at the outlet, which is also strongly related to the centrifugal impeller inlet backflow. The outlet vortex causes energy loss and total pressure reduction in this region, while the uneven velocity distribution results in higher velocities at low-pressure locations, making cavitation likely. Consequently, a cavitation region forms near the hub at the inducer outlet.

Since cavitation occurs in portions of the inducer outlet and vortices exist at the inducer exit, both severely affecting centrifugal impeller cavitation performance, cavitation also occurs at the inlet of the centrifugal impeller's long blade suction

surface.

3 Unsteady Cavitation Results

To detect pressure fluctuations in the flow field, 20 monitor points were arranged as shown in Figure 9 [Figure 9: see original paper]. Points 1-4 are uniformly distributed upstream of the inducer blades, points 5-8 are uniformly distributed between the inducer outlet and centrifugal impeller inlet, and points 9-20 are uniformly distributed inside the volute.

Fast Fourier Transform (FFT) was applied to the pressure fluctuations, with the resulting frequency domain plots shown in Figure 10 [Figure 10: see original paper]. The dominant frequency of pressure fluctuations upstream of the inducer is the inducer blade passing frequency. The dominant frequency between the inducer and centrifugal impeller is also the inducer blade passing frequency, with large amplitude variation in the circumferential direction. The secondary frequency is three times the inducer blade passing frequency, while the centrifugal impeller blade passing frequency also influences pressure fluctuations in this region, though with smaller variation. Thus, inducer blade rotation significantly impacts pressure fluctuations at the inducer outlet. Pressure fluctuations at the volute outlet positions contain frequencies corresponding to inducer and centrifugal impeller blade passing frequencies and their second harmonics. The results show large amplitude variation for the $4\times$ rotational frequency component in the volute, with monitor point 16 (located near the tongue) exhibiting high amplitude at this frequency. Additionally, large amplitude variation occurs for the centrifugal impeller blade passing frequency due to rotor-stator interaction between the rotating impeller and volute.

4 Conclusions

The ZGB cavitation model can effectively predict cavitation flow in turbopumps. The primary cavitation regions in the turbopump occur at three locations: the inducer blade inlet edge, the inducer outlet near the hub, and the centrifugal impeller blade inlet. The turbopump analyzed in this study still has room for improvement, which could be achieved by modifying the inducer inlet and outlet angles as well as the centrifugal blade inlet angle. Pressure fluctuations upstream of the inducer are dominated by the inducer blade passing frequency. Pressure fluctuations between the inducer and centrifugal impeller are significantly affected by inducer rotation. Pressure fluctuations in the volute are substantially influenced by rotor-stator interaction.

Note: Figure translations are in progress. See original paper for figures.

Source: ChinaXiv – Machine translation. Verify with original.