Source Term Based Modeling of Rotating Cavitation in Rocket Engine Turbopumps

Claudio Lettieri, Faculty of Aerospce Engineering, Delft University of Technology, Delft, The Netherlands

Adam Vermes, Faculty of Aerospce Engineering, Delft University of Technology, Delft, The Netherlands

Long Abstract

Introduction

Large-scale liquid rocket engines require high-speed turbopumps to inject cryogenic propellants into the combustion chamber. To provide the required propellant mass flow rate and pressure rise while maintaining low system mass the turbopumps must operate at high rotational speeds, which in turn lead to the formation of vapor cavities at the inlet. Inducers are used to mitigate the amount of cavitation but can experience cavitation instabilities even when operating near design conditions. Of particular concern is rotating cavitation, which is characterized by an asymmetric rotating cavity at the pump inlet that can cause severe vibrations, breaking of the pump and loss of the mission [1]. Despite much work in the field, no general design guidelines exist to avoid the onset of rotating cavitation during the pump design phase, and its occurrence is often assessed through costly experimental testing. On the one hand, reduced order models, which neglect tridimensional effects, have limited predictive capabilities [2-3]. On the other hand, high fidelity calculations are computationally expensive and cannot be used during the early design phases [4].

This paper presents a new capability to predict the occurrence of rotating cavitation during the early design phase of the turbopump. The approach uses source terms to model cavities and hydrodynamic blockage in inviscid, single-phase numerical calculations, hence accelerating the speed of the computation by two order of magnitude compared to traditional methods. The model is validated against experimental data [5-6] and high fidelity numerical calculations [4]. The computations capture both the spatial mode shape and frequency of rotating cavitation at the expected flow coefficient and cavitation number.

1. Technical Approach and Results

Source terms are added to inviscid, single-phase, unsteady, full annulus, numerical calculations to model the effects of hydrodynamic and cavity blockage on the flow in rocket turbopumps. The three-dimenaional boundary displacement thickness is modelled through a transpiration velocity at the blade surface derived from fundamental boundary layer theory [7]. The effect of cavities on the blade loading and flow blockage is modelled through source terms to the continuity and momentum equations, based on the cavitation compliance and mass gain fractor determined through first principles analysis. The source terms are casted in the form of polynomials of flow coefficient and cavitation number, maintaining a link with the instantaneous flow field during rotating cavitation.

The model is first validated with canonical two-dimensional simulations of the cavitating flow of a hydrofoil [6] and an inducer cascade [3]. The aim of the work is to apply the methodology to the study of rotating cavitation in a rocket turbopump [5]. The comparison with the hydrofoil and cascade indicates agreement of the predicted cavity blockage within 5% of the experimental value, while viscous blockage is captured with a 0.26% error. The source term modeling yields prediction of the pump performance within 10% of the measured values, further corroborating the model implementation.

Figure 1 illustrates the onset and propagation of rotating cavitation in a three-bladed inducer cascade. The inducer geometry is given by Iga et al. [2], together with measurements of the cavity volume and frequency of propagation of the instability. The calculations show close coupling of the flow dynamics on adjecent blades and indicate a periodic change in blade loading (Figure 1 top) with a frequency equal to 1.24 the rotor frequency, corresponding to rotating cavitation. The frequency is close to the value reported by Iga et al.[2] of 1.21.

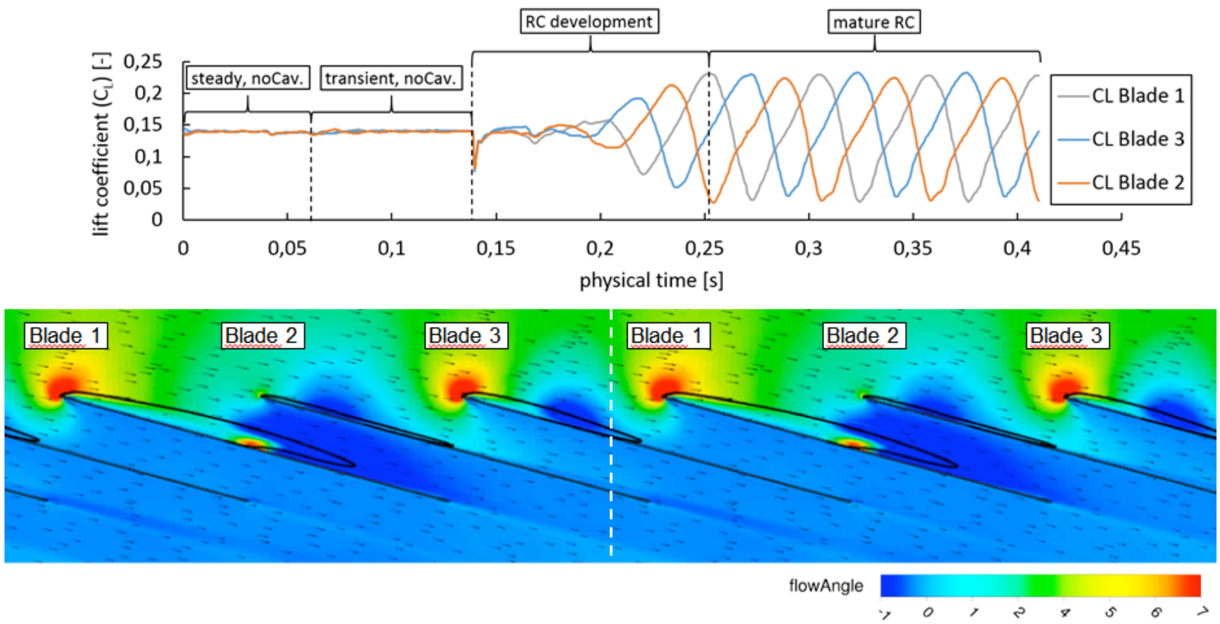

Figure 1. Numerical simulation showing rotating cavitation in three bladed inducer. The time history of lift coefficient of each blade show the development of the instability (Top). The countour plot (bottom) show the corresponding flow angle at a given instant in time.

The bottom of Figure 1 shows a snapshot of the flow field at a given instant in time. The contours indicate the flow angle, while the region of the flow where blockage terms will be implemented are indicated by black lines. Rotating cavitation is caused by the coupling of cavities on adjecent blades. When the cavity is large enough (about 65% of the blade pitch) the re-entrant jet at the back of the cavity interacts with the next blade, causing reduced flow angle and collapse of the next blade cavity. This is consistent witht he mechanism observed by Tani et al. [3], suggesting that the model can capture the relevant physical mechanism associated with rotating cavitation. Next, the source terms base model is being applied to the study of the three-dimensional flow in a four bladed inducer [5]. The model ability to capture the inducer performance and dynamic behavior is being assessed through comparison with experimental data. The scope of this work to establish a new capability to guide the design of the inducer and related casing treatment to suppress rotating cavitation instabilities.

References

[1] Brennen., C., *"Hydrodynamics of Pumps"*, Concepts ETI Inc. and Oxford University Press, Oxford, 1994, pp. 123-163.

[2] Iga, Y. and Yoshida, Y., "Mechanism of Propagation Direction of Rotating Cavitations in a Cascade," *Journal of Propulsion and Power*, Vol. 27, No. 3, 2011, pp. 675-683. doi: 10.2514/1.49454

[3] Tani, N. , Yamanishi, N., and Tsujimoto, Y., "Influence of Flow Coefficient and Flow Structure on Rotational Cavitation in Inducer," *Journal of Fluids Engineering,* Vol.134 , No. 2, pp. 21-30, 2012. doi:10.1115/1.4005903

[4] Hosangadi, A., Ahuja, V. and Ungewitter, J.R., *"Simulations of Rotational Cavitation Instabilities in the SSME LPFP Inducer"*, 43rd AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, Paper No. AIAA-2007-5536, 2007.

[5] Lettieri, C., Spakovszky, Z., Jackson, D., and Wang, V.. "Characterization of Rotating Cavitation in a Four Bladed Inducer", 52nd AIAA/SAE/ASEE Joint Propulsion Conference, AIAA Propulsion and Energy Forum, (AIAA 2016-4986)

[6] Cervone, A., Bramanti, C., Rapposelli, E., d'Agostino, L., "Thermal Cavitation Experiments on a NACA 0015 Hydrofoil", ASME, 2006

[7] Lighthill, M., J., "On displacement thickness", Journal of Fluid Mechanics, vol4(4),pp.383-392, 1958