Identifying Cavitation Instabilities in an Inducer Using Hot Water as a Cryogenic Simulant

David Jackson, Propulsion Department, The Aerospace Corporation, United States of America, Juliett Davitian, Propulsion Department, The Aerospace Corporation, United States of America, John A. Schwille, Fluid Mechanics Department, The Aerospace Corporation United States of America, Zoltan Spakovszky, Department of Aeronautics and Astronautics, Massachusetts Institute of Technology, United States of America, Claudio Lettieri, Department of Aeronautics and Astronautics, Massachusetts Institute of Technology, United States of America, and Vincent Wang, Department of Aeronautics and Astronautics, Massachusetts Institute of Technology, United States of America

SYMPOSIA ON ROTATING MACHINERY ISROMAC Since 1985

Long Abstract

Introduction

Liquid rocket engine (LRE) turbomachinery routinely operates under cavitating conditions. A variety of complex cavitation phenomena exist, which produce both broadband and discrete frequency excitation that can damage both turbopump and vehicle components. These phenomena often go unidentified during engine development and qualification due to limited predictive capability and complex dependence on engine and propellant system configuration. To address these risks, scaled room-temperature water testing and computational fluid dynamic (CFD) codes provide development tools for evaluating the impact of cavitation on pump performance and dynamic environments. While these tools produce useful data, the majority of the data and computational results apply only to inertially limited cavitation. Cryogenic fluids used on LRE turbopumps exhibit thermally limited vapor production, which can have a large effect on both pumping performance and cavitation instabilities.

Fortunately, recent work has demonstrated that hot water also experiences thermally limited cavitation and can potentially serve as a simulant for cryogenic fluids [1]. The research presented here takes advantage of hot water as a simulant for two purposes. First, the spatial structure and propagation velocity of cavitation instabilities are characterized with experiments on an inducer across a range of water temperatures selected to cover the spectrum from room temperature water to cryogenic conditions. Second, the role of the cavitation sub model in CFD is discussed. An inertially limited model is included for reference, but the focus is on two thermal models. One requires the inclusion of the energy equation in the CFD solver, while the other eliminates that need by reducing the local vapor pressure to account for the energy used in vaporization. The models' calculations are compared against data from the experiments on the inducer.

1. Methods

The experimental portion of the research is conducted at the Aerospace Corporation Cavitation Test Facility, shown in Fig. 1. This facility is a closed loop water test facility with the capability to measure steady and unsteady performance data for inducers. The effects of thermally limited cavitation are investigated by using heated water as a scaled substitute for cryogenic liquids. The facility is sized for a nominal 76.2 mm (3 inch) test article, and is designed for speeds up to 6000 rpm. The apparatus uses a stainless steel test section that incorporates measurement planes distributed between one diameter upstream and one diameter downstream of the inducer leading edge tip location. Each plane is instrumented with circumferential arrays of static and/or dynamic pressure transducer ports to allow detailed interrogation of the cavitating flow field. The test apparatus heaters can operate at temperatures up to 127 ^{o}C, providing data for thermally limited cavitation in rotating machinery that are not readily available in the literature.

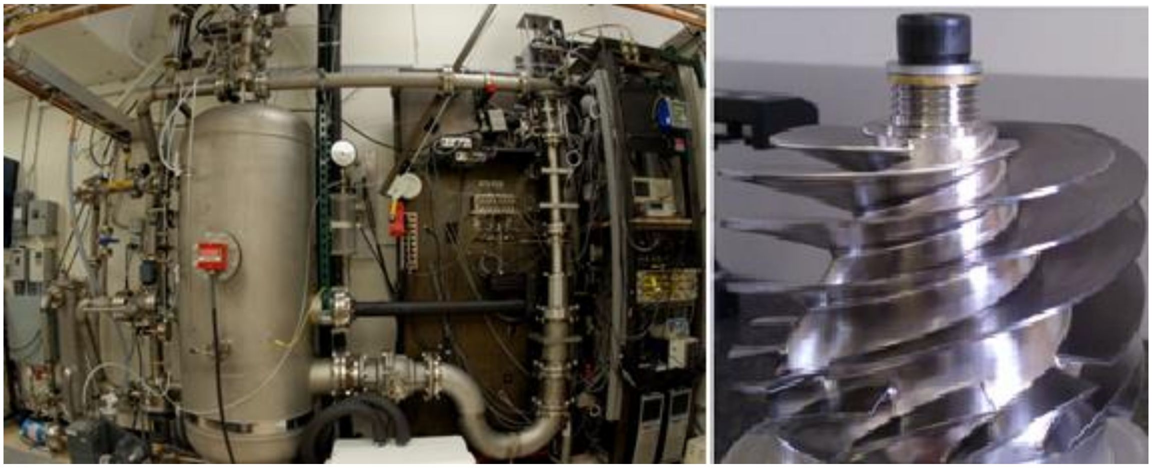

Figure 1. The Aerospace Test Facility and Test Inducer

To explore the inclusion of thermal effects on cavitation models, the commercial code ANSYS-CFX is used to solve the conservation of mass, momentum, and energy equations. By default, the code uses a cavitation submodel based on the Rayleigh-Plesset equation but neglects the thermal terms. Although this inertial model for cavitation may be acceptable for cold water, another approach is necessary for the thermally limited regime. We consider two methods to including thermal effects. In one, the CFD code includes the conservation of energy and considers the vapor pressure in the cavitation sub model to be a function of the local temperature. The other thermal model, which is based on work by Tsuda et al. [2], modifies the vapor pressure used in the cavitation sub model without calculating the temperature field by solution of the energy equation.

2. Results

The performance and dynamic data presented provide an anchoring data set as well as a comparison between thermally and inertially limited cavitation for the four bladed test inducer. The steady state suction performance data were collected at 23.3 ^{o}C, 83 ^{o}C, and 117 ^{o}C. This temperature range spans behavior from inertially limited cavitation to a regime that approaches the thermally limited cavitation in a typical LOX pump [1]. The data show a clear thermal suppression effect for steady state suction performance, where head fall off occurs at lower cavitation numbers for water at high temperature when compared to behavior in room temperature water. The dynamic data, taken at a flow coefficient of 0.08 and over a range of cavitation numbers, identify forward and backward propagating rotating cavitation modes, as well as alternate blade cavitation instabilities at multiples of 1.5, 2.5, and 2 times the synchronous shaft speed, respectively.

CFD simulations of both cold and heated water flow in the inducer also show a clear thermal suppression effect. In addition, previous research has shown that rate constants in cavitation submodels must be calibrated to generate calculations that match experimental results [3]. Here, a theoretical scaling for the rate constants in the cavitation sub models is shown to provide well matched steady state inducer performance calculations at the three temperatures. Transient simulations show indications of rotating cavitation at a flow coefficient of 0.08 and cavitation number of 0.035, which matches the experimental conditions that produce cavitation instabilities.

References

[1] Ehrlich, D. A., and Murdock, J. W., “A Dimensionless Scaling Parameter for Thermal Effects on Cavitation in Turbopump Inducers,” *J Fluid Eng,* Vol. 137, No. 4, 2015:041103.

[2] Tsuda, S., Tani, N., and Yamanishi, N. “Development and Validation of a Reduced Critical Radius Model for Cryogenic Cavitation,” *J Fluid Eng*, Vol. 134, No. 5, 2012:051301.

[3] S. Thakur, J. Wright, and C. Segal, “Development of Advanced Cryogenic Cavitation Modeling Capability Aided by Experimental Validation,” 53rd AIAA Aerospace Sciences Meeting, 5-9 January 2015, Kissimmee, FL.