CAV2021
11th International Symposium on Cavitation
May 10-13, 2021, Daejon, Korea

Cavitation Prediction in the LUMEN LCH4 Pump

Christopher Groll [12*], Tobias Traudt [2], Michael Oschwald [23] and Stefan Schlechtriem [2]

[1]Institute of Space Systems, University of Stuttgart, Germany
[2]Institute of Space Propulsion, German Aerospace Center (DLR), Germany
[3]Institute of Jet Propulsion and Turbomachinery, RWTH Aachen University, Germany

Abstract: This paper presents efforts made to predict cavitation within the centrifugal propellant pumps of the LUMEN (Liquid Upper stage deMonstrator ENgine) open expander cycle breadboard rocket engine. Specifically, the occurrence of cavitation within the fuel pump is examined. This is of interest as cavitation influences both the operating behavior of the pump as well as its operational safety. To facilitate the definition of an appropriate inlet pressure in upcoming tests and enhance the turbopump design capabilities at DLR, the functionality of the Ansys CFX flow solver is extended to account for the thermodynamic effect which significantly influences cavitation in liquids close to the critical point. The implemented model is validated against experimental data from literature and good agreement is shown. The cavitation model is applied to flow simulations of the LCH4 pump for its design operating point. It is found that the total inlet pressure must not decrease below 7 bar in order to limit cavitation while ensuring the generation of the head required by the engine.

Keywords: Turbopump; Cavitation; Liquid Methane; Rocket Engine; LUMEN;

1. Introduction

The development of the LUMEN experimental rocket engine has been driven forward at DLR Lampoldshausen for several years now [1, 2]. While combustor tests of this 25 kN thrust expander bleed cycle engine are currently being conducted, hardware manufacturing is underway for the first tubopumps designed and built at DLR. Up to this point the design of these pumps has been driven by 0D and 1D methods which are largely based on empirical relations. Resulting designs are then checked for compliance with system requirements concerning efficiency, generated head and off design operating behavior using single phase CFD calculations. Suction performance has been investigated up to this point by the 0D methods of Ovsyannikov, Brennen and Gülich [3,4,5]. Based on these investigations, a static inlet pressure lower than 6 *bar* shall be avoided during operation of the methane pump without inducer. Since LUMEN engine will be operated on the newly established P8.3 test bench, this is easily accomplished by increasing the supply pressure to the turbopump. However, this is not representative of realistic inlet pressures, which are generally found to be in the range of 3 *bar* in order to minimize structural mass of the rocket [6]. In order to produce pump designs which may include inducers and are more representative of flight hardware, while also facilitating the development of improved impeller geometries, efforts are being made to improve the available design methods by investigating the simulation of cavitation in rotating machinery. First efforts in this field are presented in this paper.

* Corresponding Author: Christopher Groll, christopher.groll@dlr.de

2. Cavitation Model and Numerical Setup

While assuming homogeneous equilibrium between the two phases, the influence of evaporation related temperature depression on local saturation pressure is modeled via an empirical relation as proposed by Utturkar and also implemented by Singhal and Tsuda as shown in equation 1 [7, 8, 9].

$$p_{sat,mod} = p_{sat,ref} - \frac{\alpha}{1-\alpha}\frac{\rho_v}{\rho_l}\frac{L}{c_{p,l}}G_{sat} + 0{,}195\rho_m k \quad (1)$$

Here $p_{sat,ref}$ is taken to be the saturation pressure at pump inlet conditions while ρ_l, ρ_v and $c_{p,l}$ denote liquid, vapour densities and specific heat of the liquid phase. The temperature depression caused by the expenditure of latent heat L during evaporation is included with the second term on the righthand side of equation 1. The Temperature depression associated with evaporation is considered using the definition of Stepanoffs B-Factor and its relation to the void fraction α for ideal fluids described by Utturkar [7, 10]. It is related to the saturation pressure by multiplying the term with the local slope of the saturation curve G_{sat}. Here it is chosen to generate this value by linearizing the slope of the saturation curve over the range of $\pm 3\ K$ with regards to the inlet static temperature in order to cover the expected range of temperature changes with some margin. Note that the vapour is assumed to always be in a saturation state with reference to the surrounding liquid temperature. The employment of the void fraction makes the introduction of a limit for cells with high vapour content necessary. For the presented results a lower limit for the term is introduced. Here 0,5 bar is selected as this value lies beneath any vapour pressure encountered in the converged solutions of the cases considered. This limit had to be determined by trial and error. Encountering a singularity for cells completely filled with vapour is prohibited via a case structure, again limiting the values which can be generated by this term to the aforementioned 0,5 bar.

An additional influence on the saturation pressure is the presence of small eddies which might promote cavitation. This is accounted for, as proposed by Singhal, with the rightmost term in equation 1 where ρ_m describes the mixture density and k the turbulent kinetic energy [8]. Again, it is found that the implementation of a limit for the contribution of this term can lead to a more robust simulation. Here it is chosen to limit the contribution of this term to two times the reference saturation pressure. This limit is again the result of a trial and error process with regards to simulation convergence behavior. The influence of this limit with regards to URANS calculations needs to be considered.

The mass transfer between the phases is described with the Schnerr-Sauer model, implemented in CFX via user functions [11]. Specific to this case, the coefficient of evaporation is set to $C_e = 0{,}01$ while, in accordance with Singhal, the coefficient of condensation is defined as $C_c = 0{,}5\ C_e$. Furthermore, instead of a fixed nucleation site number density, the bubble radius is fixed to $R_B = 1^{-6}\ m$ [8].

2.1. Numerical Setup

All presented calculations are conducted with the RANS solver Ansys CFX. The fluids are assumed to be incompressible, isothermal, immiscible and ideal. In both the simulation of the flow through the converging-diverging nozzle and the pump the k-w-SST model is used to include the effects of turbulence on the flow. This choice is made as it is both applicable to the representation of separation due to adverse pressure gradients which promote accurate cavity length depiction, as well as the ability to accurately predict the performance of turbomachinery [12].

* Corresponding Author: Christopher Groll, christopher.groll@dlr.de

A mesh sensitivity study has been performed for the pump meshs [1]. Mesh quality concerning skewedness, expansion ratio and orthogonality is ensured. A $y+\approx 1$ is targeted in the mesh generation for the bulk flow. Ansys' automatic near-wall treatment, which switches between wall functions and low-Re formulations based on mesh resolution, is utilized to try and reproduce boundary layer development. The calculations are started with the generation of a well converged cavitation free solution. Here the state of convergence is determined via mass flow and pressure monitors on the inlet and outlet boundaries as well as the standard residual outputs of the solver concerning momentum, turbulence properties and pressure. Pump simulations are conducted for a single blade passage connected to inlet and volute via mixing planes. A better overview of the LCH4 pump is given by Traudt et al. in [1, 2]. For the validation calculations total pressure at the inlet and static pressure at the outlet are prescribed. Inlet total pressure and outlet mass flow are set for the pump simulations. All walls are considered to be of the no slip type.

3. Results

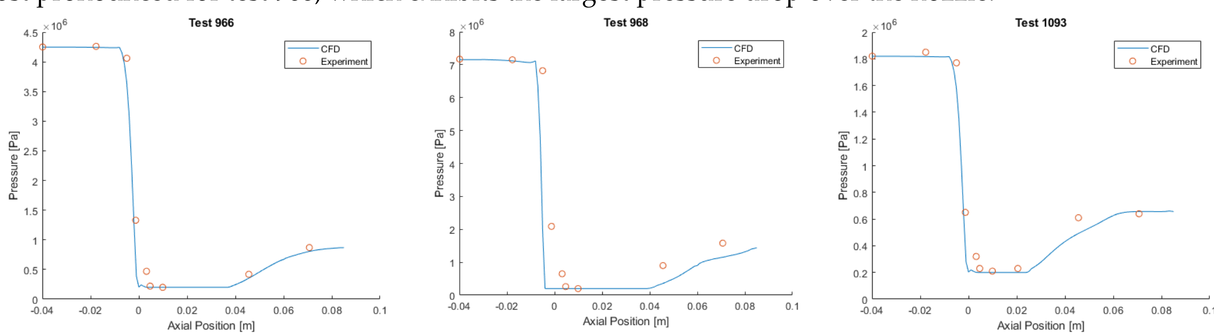
Firstly, the results of the validation are discussed. In a next step, the cavitation curve for the investigated fuel pump is generated by conducting 5 steady state simulations and varying the inlet total pressure in 1 bar steps from 10 bar to 6 bar.

3.1. Validation of Cavitation Model

The cavitation model is applied to the case described by Simoneau where a cavitating flow of methane is examined by means of pressure measurements along the wall of a converging diverging nozzle at different distances to the nozzle throat [13]. Out of the conducted experiments the three test cases also utilized by Tsuda are selected for validation [9]. While this allows for a direct comparison with the original formulation by Tsuda, the selected experiments also represent pressure levels relevant to the pump being investigated here.

Figure 1 shows a comparison of the measured and simulated pressure distributions along the nozzle wall. The x-axis indicates the distance between measurement position and the throat, so the onset of the divergence, of the nozzle. In the experiments a single pressure sensor was mounted at the wall at each examined axial position. As the validation computations are implemented as 3D simulations, the pressure distribution over the nozzle wall circumference at equally spaced intervals is averaged and plotted in figure 1.

It can be seen that good agreement is reached, which indicates that the extend of the cavitation zone can be approximated to that found in the experiments. Deviations between experiment and simulation are most pronounced for test 968, which exhibits the largest pressure drop over the nozzle.



(a) (b) (c)

Figure 1. Simulation results alongside experimental data for the tree selected validation cases.

* Corresponding Author: Christopher Groll, christopher.groll@dlr.de

3.2. Pump Simulations

In this section the cavitation model is used for the pump simulations. Figure 2 shows the head generated by the pump over the cavitation parameter, which is calculated from the static pressure at the pump inlet $p_{in,s}$ and the inlet circumferential velocity u_1, as defined in equation 2.

$$\sigma = \frac{p_{in,s} - p_{sat,ref}}{0{,}5\rho_l u_1^2} \tag{2}$$

The NPSH3 requirements of the evaluated pump design are determined by applying the methods proposed by Brennen, Gülich and Ovsyannikov are summarized in table 1. Here it has to be mentioned that the Brennen model does not include an estimate for NPSH3 but provides an estimate of the inlet pressure at which cavitation inception takes place. This is 6 bar for the investigated pump. These values are compared to the cavitation curve derived from the conducted flow simulations in figure 2.

Table 1. NPSH3 estimated generated using empirical relations.

Method	$\boldsymbol{NPSH3}$ $[\boldsymbol{bar}]$	$\boldsymbol{p_{inception}}$ $[\boldsymbol{bar}]$
Brennen	N/A	6
Gülich	5,7	32
Ovsyannikov	3,5	17

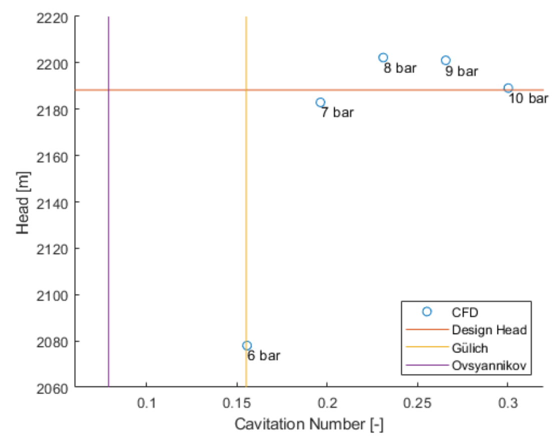


Figure 2. Calculated head generated for each evaluated inlet pressure is represented by circles. Note that the total inlet pressure is varied in 1 bar steps between the shown data points, as indicated by the associated labels. Therefore, the rightmost point represents the calculated head for 10 bar total inlet pressure. The results of the empirical estimations are shown as solid vertical lines. Gülich to the right and Ovsyannikov to the left. For reference the pump design head is indicated by a horizontal line. With the requirement that the generated head shall not drop by more than 3% from the design head, the inlet pressure must be set to 7 bar by the test bench.

4. Conclusions

It is shown that the cavitation model is able to reproduce experimental results for cryogenic fluids, in this case liquid Methane in terms of pressure levels as well as cavitation zone extent. Furthermore, it is shown that the simulation of a single cavitating passage, using Tsuda's formulation of the reduced saturation pressure, produces cavitation curves which agree with the empirical models used in the pump design up to this point. On this foundation the design of a pump which includes an inducer is started.

* Corresponding Author: Christopher Groll, christopher.groll@dlr.de

References

1. Traudt, T.; Santos Hahn, R. Status of the Turbopump Development in the LUMEN Project. 69th International Astronautical Congress, Bremen, Germany, October 2018.
2. Traudt, T.; Santos Hahn, R. LUMEN Turbopump – Design and Manufacturing of the LUMEN LOX and LNG Turbopump components. 32nd International Symposium on Space Technology and Science, Fukui, Japan, June 2019.
3. Ovsyannikov, B.; Borovsky, B. I. Theory and calculation of aggregate supply of liquid rocket engines (in Russian). Ministry of higher education, USSR, **1986.**
4. Brennan, C. Hydrodynamics of Pumps; Concepts NREC and Oxford University Press, 1994.
5. Gülich, J. F. Centrifugal Pumps; Springer Berlin Heidelberg, Germany, 2010.
6. Sobin, A. J.; Bissell, W. R. Turbopump Systems for Liquid Rocket Engines. NASA Special Publication 8107, **197**4
7. Utturkar, Y.; Wu, J. Recent Progress in Modelling of Cryogenic Cavitation for Liquid Rocket Propulsion. Progress in Aerospace Sciences, **2005**, Vol. 41(7), pp. 558-608.
8. Singhal, A. K.; Athavale, M. M. Mathematical Basis and Validation of the Full Cavitation Model. J. Fluids Engineering **2002**, Vol. 124(3), pp. 617-624.
9. Tsuda, S.; Tani, N. Development and Validation of a Reduced Critical Radius Model for Cryogenic Cavitation. J. Fluids Engineering **2012**, Vol. 134, 051301-1-10
10. Stepanoff, A. J. Cavitation Properties of Liquids. J. Eng. Power **1964**, Vol. 80, pp. 195-200
11. Schnerr G. H.; Sauer J. Physical and Numerical Modeling of Unsteady Cavitation Dynamics. Fourth International Conference on Multiphase Flow, New Orleans, USA, May 2001
12. d' Agostino, L.; Salvetti, M. V. Cavitation Instabilities and Rotordynamic Effects in Turbopumps and Hydroturbines; Springer International Publishing, Switzerland, **2017**, pp. 179-233
13. Simoneau, R. J.; Hendricks, R. C. Two-Phase Choked Flow of Cryogenic Fluids in Converging-Diverging Nozzles. NASA Technical Paper 1484, **1979**

* Corresponding Author: Christopher Groll, christopher.groll@dlr.de