Cavitation Simulation of Automotive Torque Converter Using a Homogeneous Cavitation Model

Satoshi Watanabe, Department of Mechanical Engineering, Kyushu University, Fukuoka, Japan
Keisuke Tsutsumi, Graduate School of Engineering, Kyushu University
Shin-ichi Tsuda, Department of Mechanical Engineering, Kyushu University
Takeshi Yamaguchi, AISIN AW Co Ltd., Anjo, Japan

Long Abstract

Introduction

Because of more densely packaged engine room of recent well-designed automobiles, the hydraulic torque converter, which transfers power from the engine to the transmission, is being required more compact with flattened toroidal shape, leading to more susceptibility of cavitation. The occurrence of cavitation in automobile torque converter is unfavorable in terms of noise emission [1], risk of cavitation erosion as well as deteriorated dynamic response. In our previous study [2], the naked-eye visualization of cavitation has been carried out at a stall condition with zero turbine speed using the transparent model of a typical type of torque converter. In the present study, numerical simulation of cavitation using a commercial computational fluid dynamics (CFD) code ANSYS-CFX is carried out at various speed ratios, to understand the static characteristics of torque converter against cavitation.

1. Methods

Figure 1 shows the analytical model used in the present study. Stady simulation is carried out considering only one flow passages of pump, turbine and stator elements, while the numbers of blade are 31, 29 and 20 respectively. The total number of nodes contained in this model is around 1,090K. The stage interface is applied for all the interfaces among the domains of above three elements. k-ω based SST model is employed for the turbulence model. The flow inside torque converter is basically a closed troidal flow, while, in order to set the base pressure for cavitation simulation, a part of shell wall between the exit of the pump and the inlet of the turbine is opened, where the opening boundary conditon with the specified static pressure is applied. Then, the operataing cavitation condition is expressed by using the following cavitation number based on this opening static pressure p_∞ as follows,

$$\sigma = \frac{p_\infty - p_v}{\rho U^2/2}$$

where p_v denotes the saturated vapur pressure of working fluid ATF, and ρ and U the fluid density and the peripheral speed of pump element.

Cavitation model employed here is a so-called simplified Rayleigh-Plesset model, where the production and destruction terms of void fraction is modeled based on the bubble growth driven by the diffrence between the local fluid pressure and the saturated vapor pressure. However, since the working fluid of torque converter, the automatic transimssion fluid (ATF), is known to contain large amount of air in the dissolved form, so that the observed cavitation is rather gaseous than vaporaous. Therefore, in this study, the assumed gas pressure p_g as follows is emplyed instead of the saturated vapor pressure in the Rayleigh-Plesset model.

$$p_g = \kappa HY$$

where H is the Henry's constant and Y is the mass fraction of air dissolved in the liquid (ATF). The product of HY corresponds to the equilibrium pressure of dissolved air. In the present simulation, it is set to 30% of ambient absolute pressure, which is identical to that in the dearation condition in our previous experiment [2]. The coefficient κ is unknown constant which takes some value between 0 (no air in cavitaion bubbles) and 1 (Fully filed air in bubbles).

2. Results and discussions

Figure 2 (a) shows the pump torque plotted against the cavitation number at stall condition. In addition to the present CFD simulation in two extream cases with κ=0 and 1, the experimental result [2] is also plotted for the comparison. The both CFD and experimental torques are normalized by the those in non-cavitating conditions, $T_{p-noncavi}$. The difference of $T_{p-noncavi}$ between CFD and EXP results is less than 1% in the present case. Generally, the cavitaton develops gradually with the decrease of cavitation number, and the pump torque starts to decrease with a certain amount of cavitation at some cavitation nubmer. Although the decrease of torque starts at the diffrent cavitation numbers between the present CFD and the experiment, the shape of torque drop curve is very similar. Perhaps, κ=0.5 gives us a better agreement with the experiment, although we cannot present any good explanation for this value. Figure 2(b) shows the simulated cavitation around 20% torque drop point by the iso-surface of 10% void fraction. The cavitation in the pump element at this cavitation number is very limitted, whereas the significant amount of cavitation can be observed, which is similar to our previous experimental observation [2]. According to the present CFD simulation, the troidal mass flow rate in this condition is decreased up to 15% of that in the non-cavitation condition. Therefore, the reason for the decrease of pump torque lies in the decrease of the flow rate due to the increased resistance of cavitating stator rather than the cavitation in the pump itself.

Currently, we are investigating the cavitation characteristics at other turbine/pump speed ratios for the torque converters with two diffrent stator design, which are hopefully reported in our full paper.

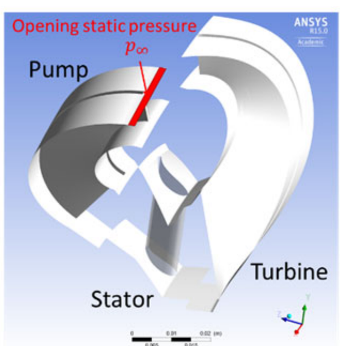
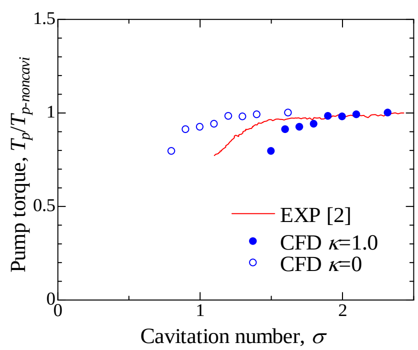
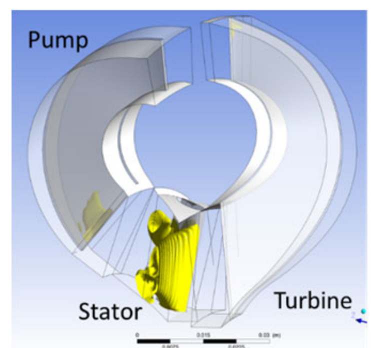


Figure 1. Analytical model



(a) Pump torque characteristics



(b) Cavitation around 20% torque drop

Figure 2. Cavitation characterstics at stall condition

References

[1] D. L. Robinette, J. M. Schweitzer, D. G. Maddock, C. L. Anderson, J. R. Blough and M. A. Johnson. Predicting the Onset of Cavitation in Automotive Torque Converters—Part I: Designs with Geometric Similitude. *Int. J. Rotating Machinery*, Vol. 2008, Article ID 803940, 2008.

[2] S. Watanabe, R. Otani, S. Kunimoto, Y. Hara, A. Furukawa and T. Yamaguchi. Vibration Characteristics due to Cavitation in Stator Element of Automotive Torque Converter at Stall Condition, *ASME Fluids Engineering Division Summer Meeting*, FEDSM2012-72418, 2012.