Flow analysis in a vane-type surface tension propellant tank

A Yu¹, B Ji¹, B T Zhuang², Q Hu², X W Luo¹, H Y Xu¹

¹ State Key Laboratory of Hydroscience and Engineering, Tsinghua University, Beijing 100084, China
² Beijing Institute of Control Engineering, Beijing 100190, China

E-mails: luoxw@mail.tsinghua.edu.cn, yua12@mails.tsinghua.edu.cn

Abstract. Vane-type surface tension tanks are widely used as the propellant management devices in spacecrafts. This paper treats the two-phase flow inside a vane-type surface tension tank. The study indicates that the present numerical methods such as time-dependent Navier-Stokes equations, VOF model can reasonably predict the flow inside a propellant tank. It is clear that the vane geometry has important effects on transmission performance of the liquid. For a vane type propellant tank, the vane having larger width, folding angle, height of folded side and clearance is preferable if possible.

1. Introduction

Surface tension tanks are widely used as the propellant management devices in spacecrafts. According to the types of propellant management devices, there are two kind of surface tension tanks: mesh typed and vane typed[1]. Mesh-type surface tension tank was mainly used in early stage. It uses mesh screen to gather and provide propellant for engine. Using mesh screen has many disadvantages, such as low strength, deformation easily, high weight and so on. Nowadays, the most widely used surface tension tanks is vane-type tank, it use vanes to manage propellant. Vane-type tank makes up the disadvantages of mesh-type one, and it can adapt to many kinds of microgravity[2].

Unlike mesh-type one, vane-type tank’s performance cannot be fully tested on the earth, it needs microgravity conditions and space-carrying experiment[3]. These tests cost highly and need numerical simulation to give a forecast results. So, during the design of surface tension tanks, numerical simulation is very important. This paper treats the two-phase flow inside a vane-type surface tension tank using VOF (volume of fluid) model. We investigated the effects of geometrical parameters of the vane, and the design optimization of the vane was discussed.

2. Propellant tank and its vane

The basic structure of the vane-typed surface tension tank is shown as figure 1. The tank is ellipsoidal, and there are several vanes for guiding liquid symmetrically distributed along the inner wall. At one end of the tank, there is a hydraulic accumulator.
In order to improve the transportation efficiency of the propellant, the vanes having two folded sides are used for guiding the propellant flow. Figure 2 shows the schematic of the vane. The vane is set along the tank inner wall, whose radius is $R$. The vane width is $L$, the height of folded sides is $h$, and the folding angle is $\alpha$. There is a clearance between the vane and tank inner wall, which is named as $c$ in figure 2. For convenience, non-dimensional parameters are defined for analysis instead of those geometries as follows:

Relative vane width i.e. $L' = L/R$.
Relative height of folded side i.e. $h' = h/R$.
Relative clearance i.e. $c' = c/R$.

In order to investigate the effects of geometrical parameters of the vane, 9 simulation models are designed and listed in table 1.

**Table 1. Simulation models for the vanes of a propellant tank.**

| Model No. | $L'$ | $\alpha$ (°) | $h'$ | $c'$ |
|-----------|------|--------------|------|------|
| 1         | 0.126| 135          | 0.0168| 0.0316|
| 2         | 0.084| 135          | 0.0168| 0.0316|
| 3         | 0.168| 135          | 0.0168| 0.0316|
| 4         | 0.126| 150          | 0.0168| 0.0316|
| 5         | 0.126| 120          | 0.0168| 0.0316|
| 6         | 0.126| 135          | 0.0126| 0.0316|
| 7         | 0.126| 135          | 0.0210| 0.0316|
| 8         | 0.126| 135          | 0.0168| 0.0210|
| 9         | 0.126| 135          | 0.0168| 0.0420|
3. Numerical methods

3.1 Governing equations

The time-dependent Navier-Stokes equations are applied to resolve the flow inside propellant tanks with different vanes. Because Reynolds number of the fluid flow is small in the tanks, the flow is treated as laminar flow.

To track the interface between the propellant and pressurized gas in the tanks, VOF model is applied with a fixed Eulerian mesh. Since the model treats two fluids as a mixture, whose physical properties such as fluid density and viscosity are time dependent and different in each computational cell, a single set of continuity and momentum equations is resolved. The volume fraction of each fluid in the tanks is obtained by resolving mass transportation equation as mentioned following.

The fundamental equations for this study are as follows:

1) Continuity equation

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho \cdot u_i) = 0
\]

where \(u_i\) is the velocity of the mixture, \(\rho\) is the density of mixture, and it can be calculated by

\[
\rho = \alpha \rho_1 + (1 - \alpha) \rho_2
\]

where \(\alpha\) is the volume fraction of the pressurized gas, \(\rho_1\) and \(\rho_2\) are the densities of the pressurized gas and propellant.

2) Momentum equation

\[
\frac{\partial}{\partial t} \rho u_j + \frac{\partial}{\partial x_i} \rho u_i u_j = -\frac{\partial p}{\partial x_j} + \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) + \rho g_j
\]

where \(g\) is the gravitational acceleration, \(p\) is static pressure, and \(\mu\) is dynamic viscosity of the mixture which can be calculated using the same method as density.

For propellant flow, surface tension makes pressure difference, i.e.

\[
\frac{\partial p}{\partial x_j} = -\sigma \frac{\partial}{\partial x_j} \left( \frac{1}{R} \right)
\]

where is \(\sigma\) surface tension.

Thus, the momentum equation becomes

\[
\frac{\partial}{\partial t} \rho u_j + \frac{\partial}{\partial x_i} \rho u_i u_j = \sigma \frac{\partial}{\partial x_j} \left( \frac{1}{R} \right) + \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) + \rho g_j
\]

3) Transportation equation

The volume fraction of the pressurized gas \(\alpha\) can be resolved by the following mass transportation equation:

\[
\frac{\partial \alpha}{\partial t} + \frac{\partial}{\partial x_i} (\alpha \cdot u_i) = 0
\]

3.2 Computation domain and mesh

Since this paper aims to evaluate the liquid transportation capability of the tank vane, the hydraulic accumulator shown as figure 1 is not considered. Figure 3 shows the computation domain, where a half of the tank and two vanes are included. For better simulation accuracy, a structured grid is applied as figure 4.
All walls are set to no penetrate and no slip wall boundary condition. On the half-cutting plane of the domain, symmetrical condition is assigned. As shown in figure 3, where a cartesian coordinate system is plotted, the direction of micro gravity is minus x.

The numerical simulation is conducted applying the solver of the commercial CFD code ANASYS. The flow inside the propellant tank is simulated for relocation at the micro gravity of $1 \times 10^{-3} g_0$, and transfer process at $1 \times 10^{-6} g_0$.

4. Result and analysis

4.1 Transfer process in micro-gravity condition

The relocation process at a micro gravity condition (i.e. $g = 1 \times 10^{-3} g_0$ at minus x direction) is simulated. During this calculation, the instantaneous mass centre of the liquid along x direction is monitored. At the instant of the completion of relocation process, a stable liquid level is observed, and the monitored mass centre of the liquid keeps constant. Figure 5 shows the snapshots of liquid levels at the instant of relocation completion for those simulation models. Note that red colour stands for the liquid, and blue stands for the gas. A reference line, where the start of liquid level for Model No.9 reaches, is drawn in figure 5.

It is seen that the liquid basically transfers in the gap between the vane and inner wall of the tank due to the surface tension effect. Compared among nine simulation models, Model No.8 has the farthest of liquid transportation, and Model No.9 has the lowest. Thus, different geometrical parameters lead to different transmission performance of the vane.
4.2 Transfer process at nearly zero gravity condition
For purpose of obtaining the effects of each geometrical parameter, the transmission performance at a micro gravity of $g = 1 \times 10^{-6} g_0$ is investigated after relocation process.

Figure 6 shows the liquid distribution of each model after 30 seconds after allocation completion. For comparison, another reference line which fits the start of liquid level for Model No.9 is shown. It is noted that the liquid moves continuously along the vane towards the opposite side of the tank. It is also clear that the transferring speed of Model No.8 is the largest, and that of Model No.2 is the smallest.

Figure 7 shows liquid distribution at different time instants for Model No.2. The transferring time is counted from the relocation completion. It is noted that the liquid moves along the vane, and finally reaches to the opposite side (relative to the initial liquid level) of the tank. The liquid level at the right side of the tank initiates from a flat curve at figure 7(a), then changes from a convex curve at figure 7(c) to a concave curve at figure 7(h). After 200 second, more than 39% of the liquid has moved to the opposite half of the tank. Because the surface tension is the main force acting on the fluid at the micro gravity condition, most liquid moves together around the vanes. Thus, the vane can perform very well in the process of filling liquid to the accumulator, through which, the tank provides the propellant to the engines.
To present quantitatively the effect of each geometrical parameter of the vane on transmission performance for the tank, the mass center of all liquid in the tank in x direction i.e. $X_c$ is calculated. A rapid change of $X_c$ means a larger transmission rate. The relationships between $X_c$ and each vane geometry are shown in figure 8.

Based on those results, the following can be seen:

Basically, the transmission rate of the liquid increases with vane width i.e. $L$, as shown in figure 8(a). It is understandable that the vane with bigger vane width has larger surface area;

Because Model No.4 with the biggest folding angle has largest among three models in figure 8(b), its transmission rate is the largest;

The transmission performance for Model No.7 with highest height of folded side among three models is the best. However, the other two models seem to have the same transmission rate of the propellant;

Figure 7. Liquid distributions in several instants for Model No.2 ($1 \times 10^{-6} g_0$).
The transmission rate of the liquid basically increases with the clearance between the tank wall and the vane. It is clear that the vane with the smallest clearance i.e. Model No.8 is not suitable for the tank. Thus, for a vane type propellant tank, the vane having larger width, folding angle, height of folded side and clearance is preferable if possible.

![Graphs showing time-dependent mass center of the liquid in x direction](image)

**Figure 8.** Time-dependent mass center of the liquid in the tank in x direction i.e. $X_c$.

5. Conclusion
The present numerical methods such as time-dependent Navier-Stokes equations, VOF model can reasonably predict the flow inside a propellant tank. It is clear that the vane geometry has important effects on transmission performance of the liquid. for a vane type propellant tank, the vane having larger width, folding angle, height of folded side and clearance is preferable if possible.

Acknowledgments
This work was financially supported by the National Natural Science Foundation of China (Project Nos 51179091, 51206087), Tsinghua University (Grant No. 20111081106), and the China Postdoctoral Science Foundation (Project No. 2011M500314).

References:
[1] Li Y 2007 *Journal of Astronautics* 3 627-623 (in Chinese)
[2] Li Y, Pan H L and Wei Y M 2011 *Journal of Astronautics* 4 54-57 (in Chinese)
[3] Hu Q, Li Y and Geng Y B 2010 *Aerospace Control and Application* 3 55-58 (in Chinese)