Numerical Simulation on the Process of Supercavity Development and the Planing State of Supercavitating Vehicle

ZHOU Jing-jun, YU Kai-ping, ZHANG Guang
(School of Aeronautics, Harbin Institute of Technology, Harbin 150001, China)

Abstract: In order to understand the evolution rule of the inner pressure of ventilated supercavity in its developing process as well as the cavity stability, unsteady three dimensional numerical simulations adopting the two-fluid multiphase flow model and DES (Detached Eddy Simulation Model) turbulence model are carried out. One method based on the relative motion and mesh deformation technology is adopted to investigate the planing state of supercavitating vehicle. The results show the method can predict the developing process of supercavity and its instability characteristic as well as the planing state of supercavitating vehicles. The numerical method can be used to further investigate the planing state and give some significant conclusions.

Key words: ventilated supercavity; two fluid multiphase flow model; pitching motion

1 Introduction

Supercavity is achieved when an underwater vehicle travels at a sufficient high speed or by injecting the non-condensable gas. Even for vehicles designed to travel at natural supercavitating velocity, the drag must be firstly reduced by ventilated supercavitating to enable the vehicles to accelerate to the conditions at which the natural supercavity can be sustained. The ventilated cavitation has been proved to be a significant drag-reduction way and receives growing research attentions among CFD practitioners. From the published literature, natural cavitation has been widely studied in homogeneous multiphase model which ignores the interfacial dynamics, that is, there is assumed to be no-slip between constituents residing in the same control volume, and the rationality of using the homogeneous model has been verified quantitatively by experiments. For ventilated supercavitating, although many investigations have been done, there is few published literatures about the details of the gas leakage of supercavity. On the other hand, the research on the planing state of vehicle is very important to the stability and control of trajectory, although some related researches have been done, there are no literatures that can be referred about true planing state.

Received date: 2010-11-15
Foundation item: Supported by the major National Natural Science Foundation of China (Grant No.10832007)
E-mail: jingjun4866@163.com.
In this paper, the developing process of supercavity and the change law of inner pressure have been investigated to show the validity of the method on predicting ventilated supercavity. And then, by combining the numerical method and mesh deformation technology as well as motion equations of vehicles, the planing state is finally obtained.

2 Numerical methods

2.1 Basic governing equations

The multiphase system investigated here is assumed to be isothermal in which the densities of the fluid are functions of pressure but not the temperature. Under this assumption, only continuity and momentum equations are solved, the energy equations are not considered.

The basic approach adopted to simulate the ventilated cavitation flows consists of solving the standard 3-D Navier-Stokes equations, turbulence equations, the gas state equation and the volume fraction equation.

The continuity equation for the single phase is:

\[
\frac{\partial (\gamma_\alpha \rho_\alpha)}{\partial t} + \nabla \cdot (\gamma_\alpha \rho_\alpha V_\alpha) = 0, \quad \alpha = 1, 2
\]

The momentum equation for the single phase is:

\[
\frac{\partial (\gamma_\alpha \rho_\alpha V_\alpha)}{\partial t} + \nabla \cdot (\gamma_\alpha (\rho_\alpha V_\alpha \otimes V_\alpha)) = \nabla \cdot (\gamma_\alpha \mu_\alpha (\nabla V_\alpha + (\nabla V_\alpha)^T)) - \gamma_\alpha \nabla p_\alpha + M_\alpha + \gamma_\alpha \rho_\alpha g, \quad \alpha = 1, 2
\]

The volume fraction equation is:

\[
\sum_{\alpha=1}^{N} \gamma_\alpha = 1
\]

The gas state equation is:

\[
\rho = \rho(P, T)
\]

where \(\rho\) is the density of air, \(P\) is the local pressure, \(T\) is the temperature which keeps constant in the simulations. \(M_\alpha\) describes the interfacial forces acting on phase \(\alpha\) from other phases.

Formulas (1)~(4) are the whole governing equations.

2.2 Turbulence model

In this paper, in order to catch the detail of cavity developing process, DES turbulence model was used, and on the other hand, with considering the computational expense the SST turbulence model was used to predict vibration of the vehicle in the cavity. A simple introduction is given in the following to the both different turbulence models.

2.3 DES turbulence model

In order to improve the predictive capabilities of turbulence models in highly separated regions, Spalart proposed a hybrid approach, which combines features of classical RANS for-
mulations with elements of Large Eddy Simulations (LES) methods. The concept has been termed as Detached Eddy Simulation (DES) and is based on the idea of covering the boundary layer by a RANS model and switching the model to a LES mode in detached regions. Ideally, DES would predict the separation line from the underlying RANS model, but capture the unsteady dynamics of the separated shear layer by resolution of the developing turbulent structures. Compared to classical LES methods, DES saves orders of magnitude of computing power for high Reynolds number flows. Though this is due to the moderate costs of the RANS model in the boundary layer region, DES still offers some of the advantages of a LES method in separated regions.

2.4 Numerical resolutions

The simulations are based on three-dimensional calculations and a finite volume discretization of these equations is used. A solver of the coupled conservation equations of mass, momentum was adopted, with an implicit time scheme and multigrid technology.

The transient term is performed with a second-order implicit scheme:

$$\frac{\partial \left( \rho \Phi \right)}{\partial t} = 1.5 \rho^{n+1} \Phi^{n+1} - 2 \rho^n \Phi^n + 0.5 \rho^{n-1} \Phi^{n-1}$$  \hspace{1cm} \text{(5)}$$

where $\Phi$ stands for $u$, $v$, or $w$ and $u$, $v$, $w$ are the three velocity components.

The diffusive terms are calculated in a central manner.

The advection schemes implemented can be cast in the form:

$$\Phi_{ip} = \Phi_{up} + \beta \nabla \Phi \cdot \nabla r$$  \hspace{1cm} \text{(6)}$$

where $\Phi_{up}$ is the value at the upwind node, $\nabla r$ is the vector from the upwind node to the ip and one high resolution scheme is adopted, which uses a special nonlinear recipe for $\beta$ for at each node, computed to be as close to 1 as possible without introducing new extrema. The recipe for $\beta$ is based on the boundedness principles used by Barth and Jesperson[14].

3 Boundary conditions and physics models

3.1 Boundary conditions

Velocity components, volume fractions, turbulence intensity and length scale are specified at velocity inlet boundary and extrapolated at pressure outlet or opening boundaries. The mass inlet boundary is defined at the blowhole. Pressure distribution is specified at pressure outlet boundary and extrapolated at inlet boundaries. At walls, pressure and volume fractions are extrapolated and the
no-slip boundary condition is specified. Some details can be seen in Fig.1.

3.2 Models in simulations

In the simulations, two models are utilized which are demonstrated in Fig.2. Fig.2(a) is used to investigate the developing process of the ventilated cavity after the vehicle travels to 25m/s. And the other model is used to predict the planing state. Some details of computational conditions are in the Tab.1.

![Fig.2 The models adopted in simulations](image)

<table>
<thead>
<tr>
<th>Models</th>
<th>Ambient pressure</th>
<th>Simulating style</th>
<th>Time step</th>
<th>Simulated environment</th>
<th>Free stream velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>(a)</td>
<td>1.5 atm</td>
<td>unsteady</td>
<td>0.0005s</td>
<td>Infinite flow field</td>
<td>25m/s</td>
</tr>
<tr>
<td>(b)</td>
<td>2.5 atm</td>
<td>unsteady</td>
<td>0.001s</td>
<td>Infinite flow field</td>
<td>25m/s</td>
</tr>
</tbody>
</table>

4 The simulations results

4.1 The simulating results of developing process of ventilated supercavity

Fig.3 and Fig.4 show the developing process of ventilated supercavity and the change law of average pressure at curve 1 which is demonstrated in Fig.1(a). It can be seen that the cavity oscillates after the supercavity forms. The supercavity forms at the time of about 0.34s, the cavity shape is showed in Fig.3, thereafter, the self-oscillation occurs and the pressure in the cavity fluctuates violently.

![Fig.3 The cavity shape at different time in the developing process](image)
The shedding of cavity results in the instability of the cavity, and both phenomena interact with and affect each other. Cavity shape oscillations can occur in the cavity length and through the convection of waves on the cavity surface. The waves intersect at the rear of the cavity, leading to the shedding of cavity just like in the Fig.3 at the time of 0.604s. The natural cavitation number $\sigma_v$ is 0.8, the ventilated cavitation number $\sigma_0$ is about 0.0235, so $\beta=\sigma_v/\sigma_0=34.2645$. According to the conclusions in Ref. [15], when $1<\beta<2.645$, the cavity is stable. So, the results in this paper accords well with Paryshev’s results.

4.2 The predictions of the planing state of supercavitating vehicles

The characteristic of planing state of supercavitating vehicle is of important significance to the stability and control of trajectory. Here, we defined the planing state as with small little wet area or pitching angle and with small change amplitude of pitching angle, however. In this paper, the motion equations that control the gesture of vehicle in the longitudinal plane are derived and by adjusting the location of gravity center, the planing state is finally obtained. The initial state is with zero pitching angle and ventilated supercavity enveloped which is demonstrated in Fig.5.

4.3 The motion equations

The motion equations in the longitudinal plane are shown in the following, the coordinate system is built on the center of cavitator.

$$m\left(\ddot{u}-rv^2x_y\right)=F_x \quad (7)$$

$$m\left(\ddot{v}+ru+rx_y\right)=F_y \quad (8)$$

$$I_z\ddot{r}+mx_y\ddot{v}+mx_xru=M_z \quad (9)$$

where, $m$ is the mass of vehicle, $u$, $v$ are the translational velocities of vehicle in the body coordinate, $r$ is the pitching angular velocity. $x_y$ is the coordinate of gravity center in the direction of x in the body coordinate. $I_z$ is the moment of inertia of vehicle. $M_z$ is calculated in simulations.

The kinetic equations are as follows:

$$X_e=ucos\theta- vsin\theta, \quad Y_e=usin\theta+vcos\theta \quad (10)$$

where, $X_e$, $Y_e$ are the translational velocities of vehicle in the ground coordinate, $\theta$ is the pitching angle.

In this paper, in order to investigate the planing state of the vehicle without considering any
other velocity disturbance, we limit the center of cavitator traveling along the x direction in the ground coordinate, so we assume that: \( X_e = 25 \text{m/s}, \ Y_e = 0 \text{m/s} \), substitute them into equation (10) and combine with equation (9), we finally obtain the equation:

\[
I_z \ddot{r} = M_z \tag{11}
\]

In the following, the planing state is finally obtained by adjusting the location of gravity center \( x_g, \ Y_g = 0.43 \text{m} \) and \( x_g = 0.23 \text{m} \). In both conditions, \( I_z = 1.36 \text{kg} \cdot \text{m}^2 \), the mass flow rate is \( 0.013456 \text{kg/s} \). The initial state is the same which is shown in Fig.5.

The results are shown in Fig.6.

In Fig.7, when \( x_g = 0.43 \text{m} \), it can be seen that the vehicle has a pitching angle of nearly 8 degrees when time is 0.27s, and when the drag also has a maximal value. The final state can be seen in Fig.6 which is not allowed for supercavitating vehicles.

![Fig.5 The initial state of pitching motion](image)

![Fig.6 The final state when x_g=0.43m](image)

![Fig.7 The change process of pitching angle and drag coefficient of vehicle when x_g=0.43m](image)

From the above conclusions, it can be seen that the numerical method in this paper can successfully predict the planing state of supercavitating vehicles and the location of gravity center seriously affects the gesture of vehicle.
5 Conclusions

In this paper, the two fluid multiphase model and DES turbulence model are used to predict the ventilated supercavity.

Firstly, the developing process and the evolution law of supercavity at 25 m/s are simulated. The results show that the shedding of cavity results in the pressure fluctuation in cavity, and both phenomena affect each other. The waves intersect at the rear of the cavity, leading to the shedding of cavity. The results show the capability of the numerical method in predicting the ventilated supercavity.

Secondly, the planing state can be obtained using the method provided in this paper and the planing state of supercavitating vehicles is seriously affected by the location of gravity center.

Finally, the numerical method in this paper still should be verified by experiments. Anyway, the accuracy of the method to predict the ventilated supercavity scale and hydrodynamics has been verified by the authors in this paper.

References


超空泡发展过程研究以及对两种研究滑行力方法的评估

周景军，于开平，张 广

（哈尔滨工业大学，哈尔滨 150001）

摘要：采用两流体模型以及 DES 湍流模型对通气超空泡发展过程以及泡内压力变化规律进行了三维数值仿真。模拟了两种泄气方式：回注射流和双涡管泄气方式，并基于文中数值方法预测通气超空泡方面的能力，对两种研究航行体滑行状态的方法进行了评估。一种方法是在水槽中的定轴俯仰运动，另一种方法是类似于约束模式实验的自由俯仰运动，两种方法都采用了网格变形技术。结果表明在相同条件下，后者可以很容易得到超空泡航行体的滑行状态而前者较难获取滑行状态，尽管在水槽中前者更易实现。文中的数值方法可以用来进一步研究滑行状态并给出一些有意义的结论。

关键词：通气超空泡；两流体多相流模型；俯仰运动

中图分类号：TV131.2
文献标识码：A

作者简介：周景军（1981-·），男，哈尔滨工业大学教授，博士生；
于开平（1968-·），男，哈尔滨工业大学教授；
张 广（1983-），男，哈尔滨工业大学博士生。
Level Set Method for Simulation of Cavitating Flows

HUANG Biao, WANG Guo-yu, ZHANG Min-di, CHEN Wei-qing
(School of Vehicle and Transportation Engineering, Beijing Institute of Technology, Beijing 100081, China)

Abstract: In this study, the Level set method is presented and applied to multiphase cavitating flows. Herein, this method is applied to flow solver that fully couples the mass, momentum, and transport equation; in numerical simulation, the Level set method with a compressive discretization scheme in both time and space to minimize the smearing of the free surface at the interface is used for simulation of nature and ventilated cavity. In these computations, three cases, including the steady nature cavitating flow around the axisymmetric body and the Clark-Y hydrofoil, and the ventilated cavitating flow around the disk are conducted, compared with the results obtained by simulation with the widely-used mixture model and experimental data, it is shown that, the Level set method is presented specifically to cavitating flows that decreases the interface diffusion where needed. Such an approach is a valid method that could reduce smearing at the cavity interface and also capture the sharp interface between different phases in an efficient and precise manner.

Key words: cavitating flow; mixture model; Level set method

CLC number: TV131.32 Document code: A

1 Introduction

Cavitating and capturing the motion of the cavity interface in multi-phase fluid flows are a challenging research field of computational fluid dynamics. The motions of cavity interfaces (nature or ventilated supercavities) are important physical phenomenon in many physically interesting. A number of methods have been developed to resolve and track this interface in Computational Fluid Dynamics (CFD) simulating multifluid flows with sharp interfaces listing advantages and disadvantages each. The Level Set method and VOF model are becoming more and more popular to capture the motion of a free surface. The volume of fluid (VOF) method has been widely applied in tracking free interface since it was proposed by Hirt and Nichols [1], in the VOF method, the volume fraction function C is defined first, and the value of C for each cell corresponds to the fraction of fluid filling each cell. Here, the free interface can then be tracked using the interface reconstruction technique and advection algorithm. Important progress has been made in the VOF method by many researchers, such as the SLIC by Noh and Woodward [2], the FLAIR by Ashgriz and Poo [3], the refined reconstruction by Youngs [4], and the CIC-
SAM by Ubbink and Issa\textsuperscript{[5]}. In parallel, the proposed Level-set formulation is based on the method for interface-fluid flow by reinitializing the interface using a signed distance from the distance. The advantage of the Level set method is the maintenance of the sharp interface which in the VOF method can only be remained by the complicated reconstruction of the interface\textsuperscript{[6]}. Numerical algorithms for the Level set equation have been further developed over the past decade. Sussman et al\textsuperscript{[7]} applied the Level set method to incompressible fluid flows in their research; the Level set method is combined for solving the mass conservation equation and the momentum transport equation. Additionally, Shu Bitan et al\textsuperscript{[8]} used the Level set method into OpenFOAM capturing the free interface in incompressible fluid flows; the terminal velocity of rising air bubble in water and the detachment of a bubble from a wall are successfully simulated. For cavitating flows, the species-mass-conservation-based Level set method was valued for natural and ventilated cavitating flows.

In present work, in order to capture the distinct interface between different phases in cavitating flows, we provide the results for the validation study of the Level set method. This method is tested on several selected problems for which there are available experimental data of nature and ventilated cavitating flows. Three different test cases for modeling the steady and transient free surface tension have been considered. Special attention is paid to the advantages of this method compared with the numerical results with the other multiphase model-mixture model.

2 Physical and numerical model

The set of governing equations consists of the conservative form of the Reynolds averaged Navier-Stokes equations, plus a volume fraction transport equation to account for the cavitation dynamics, and the turbulence closure.

2.1 Mixture model

The Navier-Stokes equations in their conservative form governing a Newtonian fluid without body forces and heat transfers are presented below in the Cartesian coordinates:

\[
\frac{\partial \rho_m}{\partial t} + \frac{\partial (\rho_m u_j)}{\partial x_j} = 0 \tag{1}
\]

\[
\frac{\partial (\rho_m u_j)}{\partial t} + \frac{\partial (\rho_m u_i u_j)}{\partial x_i} = -\frac{\partial \mathbf{p}}{\partial x_j} - \frac{\partial \mathbf{t}}{\partial x_j} + \frac{\partial}{\partial x_i} \left[ \mu + \mu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] \tag{2}
\]

The mixture property, \(\phi_m\), can be expressed as:

\[
\phi_m = \phi_1 \alpha_1 + \phi_2 (1 - \alpha_1) \tag{3}
\]

where \(\phi\) can be density, viscosity, and so on. The mixture model is very simple which treats both phases symmetrically. It is appropriate for a calculation of non-disperse liquid-liquid or gas-liquid two phase flow.

2.2 Level set method with mass transfer

The free surface flow model is a homogeneous Eulerian-Eulerian multiphase model, in the Level set method, the interface between the two phases is represented by a continuous scalar
function $\phi(x, t)$, which is set to zero on the interface, is positive on one side, and negative on the other side. This way both phases are identified, and the location of the physical interface is associated with the surface $\phi=0$. The function $\phi$ is called the Level set function and is typically defined as the signed distance to the interface: i.e., $\phi = d(x, t)$ on one side of the interface and $\phi = -d(x, t)$ on the other, where $d(x, t)$ is the shortest distance from the point $x$ to the interface.

When the interface is advected by the flow, the evolution of the Level set function is given by:

$$\frac{\partial \phi}{\partial t} + u \cdot \nabla \phi = 0$$  \hspace{1cm} (4)

In the Level set fluid-fluid formulation, the density and viscosity are typically interpolated across the interface as follows:

$$\rho(x, t) = \rho_1 + (\rho_2 - \rho_1) H_\phi(\phi(x, t))$$  \hspace{1cm} (5)

$$\mu(x, t) = \mu_1 + (\mu_2 - \mu_1) H_\phi(\phi(x, t))$$  \hspace{1cm} (6)

where subscripts 1 and 2 denote the values corresponding to the two different phases, respectively. Here, $H_\phi(x)$ is a smoothed Heaviside function which is defined in Ref.[7].

When the interface between the fluids is highly curved, then the effect of the surface tension force in the multiphase simulation becomes particularly important. Surface tension acts at the interface between two fluids. Computationally, this is awkward to deal with and the role of surface tension is generally incorporated as a continuum surface force. Using the continuum surface force model, the momentum transport equation for incompressible flow is:

$$\rho \frac{\partial u_i}{\partial t} + \rho \frac{\partial (u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) + \sigma \kappa \delta(\phi) n$$  \hspace{1cm} (7)

where, the model is based on the continuum surface force model of Brackbill et al.[10] which models the surface tension force as a volume force concentrated at the interface. $\sigma$ is the surface tension coefficient, $n$ is the interface normal vector pointing from the primary fluid to the second fluid (calculated using the volume fraction gradient), and $\kappa$ is the surface curvature defined by:

$$n = \frac{\nabla \phi}{|\nabla \phi|}, \quad \kappa(\phi) = \nabla \cdot \left( \frac{\nabla \alpha}{|\nabla \alpha|} \right)$$  \hspace{1cm} (8)

For nature cavitating flows, mass transfer is performed within a volume-fraction-based Level set method:

$$\frac{\partial \rho_\phi}{\partial t} + \frac{\partial (\rho_\phi u_i)}{\partial x_i} = \dot{m}^+ + \dot{m}^-$$  \hspace{1cm} (9)

where $\rho_\phi$ is the mixture density, $u_i$ is the velocity component in Cartesian coordinates, $t$ is the time, $\dot{m}^+$ is the condensation rate and $\dot{m}^-$ is the evaporation rate. In presented study, the source term $\dot{m}^+$ and $\dot{m}^-$ denote vapor generation (evaporation) and condensation rates which are depicted in Ref.[11].
In computations, the Level set method uses a compressive discretization scheme in both time and space to minimize the smearing of the free surface at the interface. Typically, this reduces smearing at the interface to two or three layers of cells. The Level-set method is an interface capturing approach rather than an interface reconstruction approach.

2.3 Turbulence model

In present study, the SST model [12] is chosen for steady simulation and the DES model based on the SST formulation is chosen for transient simulation. The advantage of this combination is that the accurate prediction of turbulent boundary layers up to separation and in separated regions carries over from the SST model. In addition, the SST model supports the formulation of a zonal DES formulation [13], which is less sensitive to grid resolution restrictions than the standard DES formulation.

Detached- Eddy Simulation model:

\[
\begin{align*}
\frac{D \rho k}{Dt} &= \rho k^{3/2} \frac{\partial}{\partial x} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x} \right] + \frac{\partial}{\partial x} \left( \frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x} \right) + 2\rho (1 - F_1) \sigma_{\omega^2} \frac{1}{\omega} \frac{\partial k}{\partial x} \frac{\partial \omega}{\partial x} \\
\frac{D \rho \omega}{Dt} &= C_w \rho k^{3/2} \beta_k \rho \omega^2 + \frac{\partial}{\partial x} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial \omega}{\partial x} \right] + 2\rho (1 - F_1) \sigma_{\omega^2} \frac{1}{\omega} \frac{\partial k}{\partial x} \frac{\partial \omega}{\partial x}
\end{align*}
\] (10, 11)

All the coefficients are listed for completeness in Ref. [13], and the eddy-viscosity is defined as:

\[
\nu_t = \frac{a_1 k}{max(\bar{a}_1 \omega; \Omega)}
\] (12)

The length scales of the model in terms of \( k \) and \( \omega \) read:

\[
l_t = k^{1/2} / \beta_k \omega
\] (13)

The idea behind the DES model for Strelets [13] is to switch the SST-RANS model to a LES model in regions where the turbulent length, predicted by the RANS model is larger than the local grid spacing. In this case, the length scale used in the computation of the dissipation rate in the equation for the turbulent kinetic energy is replaced by the local grid spacing. Here, \( \Delta \) is based on the largest dimension of the grid cell, \( \Delta = \max(\Delta x, \Delta y, \Delta z) \).

\[
\varepsilon = \beta_k \omega = \begin{cases} \frac{k^{3/2}}{C_{DES} \Delta}; & C_{DES} \Delta < l \omega \\ \frac{k^{3/2}}{l_t}; & C_{DES} \Delta \geq l \omega \end{cases}
\] (14)

For wall- bounced separated flows, the above formulation results in a hybrid model that functions as the standard SST model inside the whole attached boundary layer, and as its sub grid-scale version in the near wake.

3 Results and discussion

3.1 Steady natural cavitation on cylindrical bodies

Rouse and McNown [14] carried out a series of experiments on natural cavitation about ax-
isymmetric configurations. For each configuration, measurements were made across a range of cavitation numbers. The hemispherical configuration is computed here, the experiments were performed at Reynolds number greater than 100 000 based on the diameter. As shown in Fig.1, the boundary conditions used in the present simulations include inflow, outflow and no-slip, standard boundary conditions for incompressible flow are applied: the velocity is imposed at the inlet and the pressure is fixed at the domain outlet. A no-slip boundary condition is used at both the upper and lower walls. Here, an appropriate grid of dimension is used, as shown in Fig.2, the O type mesh is chosen because of the rounded leading edge. Based on the diameter of the cylinder, a value of \( Re = 1.36 \times 10^5 \) is used for simulations in computations, and the diameter of the cylinder is 0.02 m.

![Fig.1 Outline of the computational domain](image1)

![Fig.2 Computational grids around body](image2)

Fig.3(a) and (b) show flowfield snapshots of the steady cavitating flow over a hemispherical axisymmetric body, the contours of volume fraction of vapor phase are shown respectively when the cavitation number is 0.3. For all cases, the presences of the bubble manifest itself as a decrease in magnitude, flattering and lengthening of the pressure minimum along the surface. Also, bubble closure gives rise to an overshoot in pressure recovery due to the local stagnation associated with free-stream liquid flowing over the convex curvature at the aft end of the bubble. But the cavity interfaces captured with different model are noticeable different, in the downstream region of the nature cavity, the cavity interface obtained by Level set method is sharper. In addition, in the cavitation area, the vapor volume fraction is moderately larger than the results obtained with the mixture model.

Fig.3(c) shows the comparisons between predicted and measured surface pressure distributions in parallel. Both the models match the experimental data satisfactorily. Differences in the performance are more noticeable in the closure region, where the vapor phase condenses. Fig.3(d) shows the corresponding density distribution along the surface. As seen from density plots, the liquid phase first expands and vapor phase appears uniformly inside the cavity, then the vapor phase is compressed, in a shock like fashion, back to the liquid phase. The differences among the two models in density profiles are significant. This implies that each model generates different compressibility characteristics, although they produce very similar steady-state pressure distributions; which model produces the correct compressibility is an open question and needs further investigation.
3.2 Unsteady natural cavitation around a Clark-Y hydrofoil

Unsteady nature cavitating flow over the Clark-Y hydrofoil is investigated using the Level set method. In such cavitating case, liquid-vapor mass transfer occurs through local fluctuations about the saturation pressure. Here, the geometry is based on the Clark-Y profiles, has a 0.07m chord, and is twisted a total of 8 degrees. The computational model setup approximates to the experiments of Wang et al.\textsuperscript{[15]}, simulating the hydrofoil within inviscid wall. In the experiment, the tunnel was run various speeds and pressures, visual comparisons are made with the results at $V_\infty=10\text{m/s}$, and $\sigma=0.8$. 

![Fig.3 Contour of volume fraction of vapor phase and density distribution with different models ($\sigma=0.3$)](image)

(a) Contour of volume fraction of vapor phase with Level set method
(b) Contour of volume fraction of vapor phase with mixture model
(c) Comparisons between predicted and measured surface pressure distributions
(d) Density distribution along the surface with different models

![Fig.4 Outline of the computational domain](image)

Fig.4 Outline of the computational domain

![Fig.5 Computational grids around a hydrofoil](image)

Fig.5 Computational grids around a hydrofoil
Fig. 6 displays comparisons of the unsteady cavity around the Clark- Y hydrofoil between the mixture model and the Level set method, the temporal evolution of the computed and experimentally observed flow structures with cloud cavitation ($\alpha=0.8$) is shown. Here, the experimental visual images are shown in Fig. 6(a), the black and white images correspond to an instantaneous visualization of one cycle, and Fig. 6(b) and (c), show the time sequences of flow structures predicted by different models, the blue color corresponds to liquid and the red color corresponds to a highly concentrated bubble flow. 

<table>
<thead>
<tr>
<th>Time</th>
<th>Unsteady Cavity Shapes</th>
</tr>
</thead>
<tbody>
<tr>
<td>$t_i + 17.5$ ms</td>
<td><img src="image1.png" alt="Image" /></td>
</tr>
<tr>
<td>$t_i + 24.5$ ms</td>
<td><img src="image2.png" alt="Image" /></td>
</tr>
<tr>
<td>$t_i + 30.5$ ms</td>
<td><img src="image3.png" alt="Image" /></td>
</tr>
<tr>
<td>$t_i + 35$ ms</td>
<td><img src="image4.png" alt="Image" /></td>
</tr>
</tbody>
</table>

There is a good agreement between the numerical and experimental results concerning the external shape and global structure of both attached cavity and vapor cloud shedding is achieved; but the vapor/water interface and vapor volume fraction with different models are distinct. The cavity got by Level set method is more obvious with larger vapor volume fraction. And also, the Level set method scheme prediction displayed the increased cavity size. This is because, for the Level set method in present study, the anti-diffusive characteristics of this scheme predict the nature cavity with much sharper, and hence it provides crisp interface for both steady-state and transient simulations. 

3.3 Axisymmetric ventilated disk cavitor

The case relevant to the cavitating flows is ventilated cavitating flows around an axisymmetric disk cavitator. In this case, liquid flow around the disk and a gas injected into the wake form supercavities. The geometric configuration is displayed below in Fig. 7, with the disk positioned such that the flat front is perpendicular to the free-stream velocity vector of a liquid flow. Such a case, where the air gas is injected into a liquid flow to form a cavity, is referred to as artificial or ventilated cavitation, the diameter of disc $D_n$ is 15mm. In numerical simulation,
the grid distribution is shown in Fig. 8.

![Fig.7 Schematic diagram and computational domain](image1)

![Fig.8 Computational grid for flow analysis](image2)

There are two cases predicted with different multiphase models, the assumed conditions for the cases are: for case-1, $U_w = 8\text{m/s}$, and $\sigma = 2.37$, ventilated rate $Q$ is $400\text{L/h}$; for case-2, $U_w = 3\text{m/s}$, and $\sigma = 22.56$, ventilated rate $Q$ is $600\text{L/h}$.

Solution comparisons with and without the Level set method are evaluated in Fig. 9. In our computation, the gas volume fraction isosurface 0.8 is defined as ventilated cavity. In both cases, without the interface sharpening of Level set method, the predicted profiles of the ventilated cavity tend to converge on a relatively smaller cavity size to the experimental visualizations and the numerical results obtained by Level set method. Whereas, the scheme with Level set method converges a cavity size shape that is quite similar to experimental photographs.

<table>
<thead>
<tr>
<th>Results Models</th>
<th>Ventilated cavity shapes</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>(a) Case-1</td>
</tr>
<tr>
<td>Experiment</td>
<td><img src="image3" alt="Experiment" /></td>
</tr>
<tr>
<td>Mixture model</td>
<td><img src="image5" alt="Mixture" /></td>
</tr>
<tr>
<td>Level set method</td>
<td><img src="image7" alt="Level set" /></td>
</tr>
</tbody>
</table>

![Fig.9 Visual comparisons of the predicted cavities with observations from experiment](image9)

Compared with the experimental data, the predicted cavity profiles at different models are examined in Fig. 10. The numerical results vividly show significant benefit in the prediction using the free surface sharpening approach, the axial cavity width ($D_a$) with Level set method are well predicted. As we know, mixture model allows mutual penetration and mixing between different phases, there is large numerical dissipation in vacuoles length and radius of the direction, particularly reflected in the length direction. But, the Level set method includes complete interface features, and with the surface tension model, it is easy to handle diffusion and
pulsation characteristics in numerical simulations.

4 Conclusions

In this present study, the Level set method for nature and ventilated cavitating flows are validated, various cases including the steady cavity around the cylindrical body, unsteady cavitating flow around the Clark-Y hydrofoil and the ventilated cavitating around a disk are presented. Compared with the results obtained by mixture model and experimental visualizations:

(1) For nature cavitating flows, although all three models give satisfactory predictions in overall pressure distributions and unsteady cavity shapes, differences are observed in the closure region of the cavity with the mixture model and Level set method, resulting from the differences in compressibility characteristics handled by each model. The cavity interface obtained by Level set method is sharper. In addition, in the cavitating area, the vapor volume fraction is moderately larger than the results obtained with the mixture model.

(2) For ventilated cavitating flows, the Level set method modified forms resulted in improved numerical accuracy and the ability to better resolve the cavity shape with the same cells. The scheme with Level set method converging a ventilated cavity size shape is very similar to experimental photographs and data, compared with the results obtained by mixture model.

References


---

**Level set 方法在空化流动计算中的应用**

**黄 彪，王国玉，张敏弟，陈维清**

（北京理工大学机械与车辆工程学院，北京 100081）

**摘要：** 文章将 Level set 方法应用于自然空化与通气空化流动计算中，通过耦合入相同的质量传输方程，建立了一个用于描述气、汽、液多相空化流动的数值模拟方法，分别对绕圆头回转体和 Clark-Y 型水翼的自然空化流动和绕圆盘的通气空化流动进行了数值模拟研究，并与实验结果进行了对比，研究结果表明：相常有的多相流模型（mixture model），Level set 方法通过时间和空间上的压缩离散方案，减小了相间界面的扩散，准确地捕捉到了相间的界面，可以有效地应用于空化多相流动。

**关键词：** 空化流动；混相模型；Level set 方法

**中图分类号：** TV131.32

**文献标识码：** A

**作者简介：** 黄 彪(1985- )，男，北京理工大学机械与车辆工程学院博士研究生；

王国玉(1961- )，男，博士，北京理工大学机械与车辆工程学院教授/博士生导师；

张敏弟(1970- )，女，博士，北京理工大学机械与车辆工程学院副教授；

陈维清(1985- )，男，北京理工大学机械与车辆工程学院硕士研究生。
Optimal Thrust Allocation Based on Fuel-efficiency for Dynamic Positioning System

YANG Shi-zhi¹,², WANG Lei², ZHANG Shen²
(¹ China Classification Society Nanjing Branch, Nanjing 210011, China; ² The State Key Laboratory of Ocean Engineering, Shanghai Jiao Tong University, Shanghai 200240, China)

Abstract: The two several methods of thrust distribution in a propulsion system of Dynamic Positioning System (DPS) are presented in this paper. With detail simulation and analysis, this paper concentrates on finding out an optimal thrust allocation (TA), which could gain desired forces and moments acting on the vehicle so as to keep position. Special attention is paid to solve the non-linear constrained thrust allocation problems. And the proposed methods are to set up a configuration matrix which exactly describe the layout of thrusters of the DP propulsion system. An illustrative example based on Genetic Algorithm (GA) and Sequential Quadratic Programming (SQP) is provided to demonstrate the effectiveness and correctness of the proposed methods. Five algorithms suited for different control conditions are presented.

Key words: Thrust Allocation (TA); Dynamic Positioning System (DPS); Genetic Algorithm (GA); Sequential Quadratic Programming (SQP)

CLC number: U664.3 Document code: A

1 Introduction

As the last potential supplier of natural resources and energies as well as provisions on the earth, the ocean is requested more and more to reveal its substances to human beings. Dynamic Positioning System is the most promising means of position-keeping for deep-sea exploration, which is not confined by depth of water and has much great advantages compared with mooring system. Besides DP could serve as an auxiliary of mooring system in shallow water condition, which could enhance the capability of mooring system. Basics of DP are shown in Fig.1.

This work is motivated by the fuel-efficiency optimal control allocation problem in the DP propulsion system, which is essential for DP. The main contribution of this work is to analyze the objective function and boundary condition based on Genetic Algorithm (GA) and Sequential Quadratic Programming (SQP). In this paper, we will try to solve the over-actuated optimal nonlinear constraint control allocation problem using the method proposed by Johansen & Fossen(2004)[1].

Received date: 2010-04-09 Revised date: 2010-09-13
Foundation item: Supported by National Hi-tech Research and Development Program (2008AA09Z315)
It is all right to introduce the two optimal methods and their application in DP. And it will be described simply how the objective function and boundary condition are built by some other authors, such as Fossen (2004) and Liang & Cheng (2004)\textsuperscript{[2]}. Some questions needed to be solved or completed, for example, Garus (2004)\textsuperscript{[3]} only considered one situation that the azimuth angles of thrusters were fixed and did not present how the thruster configuration matrix was given out, and only gave out a very simple numerical example to solve the static issues.

In generally, it is not very easy to solve nonlinear constrained optimal problems with Penalty Function Method. Actually based on Genetic Algorithm or SQP these deficiencies could be properly overcome. As a matter of fact, generalized multiplier method has great advantages in solving nonlinear optimal problems with hybrid constrained boundaries. Johansen (2003) introduced a slack variable formulation which could guarantee that the optimization problem always had a feasible solution, and had solved the singularity issue of thruster configuration matrix based on SQP.

Focusing on the essential requirements for a DP vessel that can rapidly and exactly distribute thrust and rotate the angle of thruster, this paper presents an effective optimum control of thruster system, using the SQP and GA with constraints on the azimuth thrusters of a propulsion system. Comparing these two thrust allocation methods, and two important useful methods are given how to improve them in order to carry these theories into practice. Finally based on a typical propulsion system of one semi-submerged platform, one numerical example will be elaborated in this work.

2 Statement of 3-DOF thrust allocation

In DP, only 3 surface motions are taken into consideration, which are surge, sway and yaw. Here the generalized force vector produced jointly by the actuators is given by

\[ \mathbf{\tau} = B(\alpha) \mathbf{u} \quad \text{(1)} \]

where, \( \mathbf{\tau} = (\tau_x, \tau_y, \tau_n)^T \), and its components are the surge force \( \tau_x \), sway force \( \tau_y \) and yaw mo-
ment $\tau_n$. The vector $u$ contains the magnitude of the force produced by each individual actuator, uniquely related via invertible actuator characteristics to low-level control signals such as propeller speed or pitch, or intermediate-level control signals such as shaft torque or power (Han, 1977)\cite{4}.

3 Propulsion system of DP

A marine vessel is usually equipped with a number of propulsion devices, such as main propellers, rudders, tunnel thrusters, azimuth thrusters and podded thrusters (Fossen, 2002)\cite{5}. In thrust allocation logic some typical characteristics of propulsion system require to be taken into consideration, such as the maneuverability and hydrodynamic interaction with current or hull of vessels. Actually thrust loss due to thruster–thruster interaction is proved to be very terrible, when two adjacent thrusters are parallel aligned in extreme sea conditions. Consequently one boundary of optimal thrust allocation logic named forbidden vector is added and proved to be feasible. In this paper, we will give a math model to depict the forbidden vector, as Fig.2 showing a sketch of forbidden vector. It can be seen that the forbidden angle is $15^\circ$.

3.1 Math model of thrust loss

At first we could get the open water characteristics of the thrusters through model test or numerical calculation, and both thrust coefficient and moment coefficient could be obtained as a corresponding continuous function through curve fitting method, just as follows:

\[
\begin{align*}
K_T &= \sum_{i=0}^{n} \sum_{j=0}^{n} C_T(i, j) (P_{D})^{i} J^{j} \\
K_Q &= \sum_{i=0}^{n} \sum_{j=0}^{n} C_Q(i, j) (P_{D})^{i} J^{j}
\end{align*}
\]

(2)

And then through a series of the hydrodynamic interaction model tests we could improve the Eq.(2), and give,

\[
\begin{align*}
K_T &= f(\alpha_1, \alpha_2, n_1, n_2, v_c) \\
K_Q &= f(\alpha_1, \alpha_2, n_1, n_2, v_c)
\end{align*}
\]

(3)

where, $\alpha_1$ is vector of the thruster and $\alpha_2$ is one of the other thruster, $n_1$ and $n_2$ are the rotational velocities of the two adjacent thrusters respectively, $v_c$ is current velocity. Actually it is difficult for us to give an exact math model to depict the thrust loss at different conditions, for hydrodynamic interaction problem depends on a lot of factors. So we only choose several typical conditions in our logics, for instance, the Extreme Sea State when all thrusters work at maximum velocity. In fact it is efficient and effective to choose a simple case at the threshold of one complex problem. As for exact math model of thrust loss, there is still a long way to go, which should involve many layers in the optimal thrust allocation logic in order to suit for different conditions.
3.2 Definition of forbidden vector

Take No.5 thruster as an example, supposing the forbidden angle is 15°. When both the azimuth angles of No.5 & No.1 thrusters are -21°, the two thrusters are aligned in a line, so the azimuth angle domain of -41° to -11° is forbidden. All azimuth angles are from 0° to 360°, as a result, the azimuth angle $\alpha$ of No.5 thruster has to satisfy as follows:

$$\alpha \geq \alpha_u \text{ and } \alpha \leq \alpha_l$$

(4)

Consequently we attained one inequality instead of (3),

$$g(x) = \left( x - \frac{\alpha_s + \alpha_u}{2} \right)^2 - \left( \frac{\alpha_s - \alpha_u}{2} \right)^2 \geq 0$$

(5)

In Fig.2, $\alpha_u$ equals 349° and $\alpha_l$ equals 319°.

![Fig.2 Sketch of forbidden vector](image)

4 Simplification of the math model and verification

Generally, the objectives of thrust allocation logic are the fuel-efficiency and maneuverability of the DPS, or any one of the both. Now for simplification aspects we only take the fuel-efficiency into consideration, actually the maneuverability of the system depends on many factors, such as the sea loads condition, thruster orientation and its response speed. And the maneuverability will be very different between in calm water and in current flow conditions. However, in this paper we mainly compare the two optimization methods and give out some advices for their engineering application. So we take a very simple case as an example, just only consider the fuel-efficiency objective.

For the sake of safety, we should set some terms to avoid singularity in order to improve maneuverability of DP in engineering application, such as the singularity of the system. An actuator configuration is said to be singular if it cannot produce forces/moments in every direction in the 3-dimensional space of surge force, sway force and yaw moment (Fossen, 1994).
which could be avoided through improving the configuration matrix.

4.1 Sequential Quadratic Programming

SQP approach can be used both in line search and trust-region frameworks and it is appropriate for small or large problems. Unlike sequential linearly constrained methods, which are effective when most of the constraints are linear, SQP methods show their strength when solving problems with significant nonlinearities (Johansen & Fossen, 2004).

SQP methods represent the state of the art in nonlinear programming methods. Schittkowski (1985), for example, has implemented and tested a version that outperforms every other tested method in terms of efficiency, accuracy, and percentage of successful solutions, over a large number of test problems.

Based on the work of Biggs (1975), Han (1977), and Powell (1978), the method allows you to closely mimic Newton's method for constrained optimization just as is done for unconstrained optimization. At each major iteration, an approximation is made of the Hessian of the Lagrangian function using a quasi-Newton updating method. This is then used to generate a QP subproblem whose solution is used to form a search direction for a line search procedure.

4.2 Genetic Algorithm

Genetic Algorithms (GA) are adaptive heuristic search algorithm premised on the evolutionary ideas of natural selection and genetic. The basic concept of GAs is designed to simulate processes in natural system necessary for evolution, specifically those that follow the principles first laid down by Charles Darwin of survival of the fittest. As such they represent an intelligent exploitation of a random search within a defined search space to solve a problem.

The advantage of the GA approach is the ease with which it can handle arbitrary kinds of constraints and objectives; all such things can be handled as weighted components of the fitness function, making it easy to adapt the GA scheduler to the particular requirements of a very wide range of possible overall objectives.

5 Numerical model

The location of thrusters is shown in Fig.3. The locations of 8 azimuthing thrusters are illustrated by a matrix \((a_i, b_i)\):

\[-43.2, -25\], \([-17.6, -35]\);
\[-43.2, 25\], \([17.6, -35]\);
\[(43.2, 25)\], \([-17.6, 35]\);
\[(43.2, -25)\], \([17.6, 35]\).

Objective function:

\[f(x) = \sum_{i=1}^{8} x_{2i-1}^{3/2} \]  

s.t.:
where \( F_x \) is the required transverse force; \( F_y \) is the required longitudinal force, \( M \) is the required turning moment.

In Eq.(7) \( g_1 \), \( g_2 \) and \( g_3 \) could be greater than or equal to zero, which could be modified according to requirement. In this paper, there are four cases including with or without \( g_4 \), and \( g_1 \), \( g_2 \) and \( g_3 \) could be greater than or equal to zero, which are displayed in Tab.1.

Input sea load values composed of required transverse force, longitudinal force and turning moment, which are including some few typical conditions of sea loads, are as following staircase functions. The input values of all logics in this paper are the same just as shown in Fig.4.

SQM results are shown in the following Fig.5, which is without constraints on forbidden vector of thrusters. It can be seen that the delivered thrusts and vector of the 8 azimuthing thrusters are not very different. That is to say, in condition of no forbidden vector of thrusters, the directions of all thrusters are almost the same, which is, just as our imagination, the optimal solution.
Genetic Algorithm results with constraints on the forbidden vector of thrusters in order to reduce thruster-thruster interaction are shown as follows. Forbidden angle is set as $10^\circ$, and the input values are the same as SQP.

<table>
<thead>
<tr>
<th>Results based on GA</th>
<th>inequality constraint</th>
<th>equality constraint</th>
</tr>
</thead>
<tbody>
<tr>
<td>forbidden vector</td>
<td>no</td>
<td>forbidden vector</td>
</tr>
<tr>
<td></td>
<td>no</td>
<td>no</td>
</tr>
</tbody>
</table>

There is not so much space in this paper to show all simulation results, so we take No.4 thruster and No.6 thruster as examples to illustrate the problem, the results in different conditions are as follows, shown as Figs.6-9.

**Tab.1 Four conditions of the algorithm**

![Fig.6 Results of inequality constraint with forbidden vector](image)

![Fig.7 Results of inequality constraint without forbidden vector](image)

![Fig.8 Results of equality constraint with forbidden vector](image)
5.1 Analysis of the simulation results

From the simulation results, it can be seen that the optimum solution is not only one, but there exists a set of solutions, which are all suitable for the control system and satisfy the objective function and constrained boundaries. Of the results, it is apparent that the magnitudes of fuel-consumption based on SQP are less than the ones based on GA at the same sea conditions. The reason is that the latter one takes thruster-thruster interaction into consideration and sets forbidden vector, but results of former is not and supposes that the thrust loss was small. Actually in most conditions including operational sea condition, it is true that the thrust loss is not very big, so in the thrust allocation logic it does not need to be taken into consideration. A balance should be built between thrust loss due to thruster-thruster interaction and how much effect could be delivered due to added forbidden vector. There is another question needed to be verified, that is how many thrusters work together in relatively calm sea load environment when it is not necessary for all thrusters work at the same time but only several ones are needed to keep station. Both of these two questions are not included in the paper, now we are here aiming for a feasible method to solve optimal thrust allocation problem. As for perfection and improvement of the objective and boundaries, it depends on the special application and request. And we compare the fuel-consumption at four different boundaries condition, that are inequation constrained with forbidden vector, inequation constrained without forbidden vector, equation constrained with forbidden vector and equation constrained without forbidden vector. Longitudinal coordinates in Fig.10 is the total energy consumption (P/kW) and horizontal ordinate shows the time domain from 0 to 600s. From Fig.10 it could be seen that four cases are almost accordant with each other, but energy consumption of the 3th case (equation and forbidden) is the biggest among the four cases. That is to say, forbidden vector boundary only makes sense at some special area which depends on the sea conditions. However, in most time-domain, fuel-consumption of the case with forbidden vector is relatively more than ones without no forbidden vector, and the fuel consumption of equation constrained boundaries is little more than inequation ones, which are all match with the fact.
6 Conclusions

This paper presents several logic design means for TA layer of DPS. It concentrates on finding out an optimal TA logic in order to gain ideal values of forces and moments acting on the vehicle, aiming for totally auto-control mode of DPS. Special attention is paid to how to build and simplify the constrained boundaries. One proposed means builds a configuration matrix to describe the layout of thrusters in propulsion system. And this paper analyses several algorithms of TA for different constrained control conditions. One disadvantage of SQP is that it is hard to attain a global optimal solution, which terribly depends on the initial value. So it is necessary to improve constrained boundaries using some special methods such as setting a weight term to penalize the error between the commanded and achieved generalized force, which is aiming for guaranteeing that the optimization problem always has a feasible solution (Johansen & Fossen). As for GA, it has a good feature in global convergence and does not very depend on the initial value, which could be fit for the optimal thrust allocation algorithm if properly improved in some details. Our research shows that it converges very slowly.

References


基于能耗最优化的动力定位推力分配逻辑算法研究

杨世知 1, 2, 王 磊 2, 张 申 2
(1 中国船级社南京分社, 210011; 2 上海交通大学海洋工程国家重点实验室, 上海 200240)

摘要: 文章为动力定位系统执行器的设计提供了两种推力分配最优化的求解方法, 旨在寻求一种理想的推力器输出, 包括推力与方位角, 以维持动力定位系统能力。通过求解非线性约束最优化问题, 得出时效性、稳定性与可控制性强的推力分配模块。文中采用一个或若干个合适的结构矩阵来描述推力器的位置及其性能, 采用不同的边界条件对系统要求进行描述, 如降低推力损失等。基于一个简单的算例, 分别采用遗传算法和序列二次方法进行求解, 改变约束条件并分析程序运行结果, 对比信号输出及其能耗情况, 论证两种算法的可行性与稳定性。文中还对比了五种不同的情况, 包括约束条件以及算法的变化, 以满足不同的应用需求。

关键词: 推力分配(TA); 动力定位系统; 遗传算法; 序列二次最优化方法

中图分类号: U664.3
文献标识码: A

基金项目: 国家高科技研究发展规划(863 计划) (2008AA092315)

作者简介: 杨世知(1984-), 男, 硕士, 主要从事海洋平台动力定位系统的研究, E-mail: dpsystem@yahoo.cn；
王 磊(1971-), 男, 上海交通大学副教授;
张 申(1986-), 男, 上海交通大学硕士生。
Investigation on Sloshing Effects of Tank Liquid on the FLNG Vessel Responses in Frequency Domain

ZHAO Wen-hua, HU Zhi-qiang, YANG Jian-min, WEI Yue-feng
(State Key Laboratory of Ocean Engineering, Shanghai Jiao Tong University, Shanghai 200240, China)

Abstract: FLNG is a new type of floating offshore unit, which has a ship-type FPSO hull equipped with LNG storage tanks and liquefaction plants. The coupling effects and interactions between ship motion and inner-tank sloshing are investigated with the help of the code WADAM in frequency domain. A unified approach where the interior wetted surfaces of the tanks are included as an extension of the conventional defined exterior wetted surface of the body is adopted, in which all of the tank and hull wetted surfaces form a large global boundary surface. RAOs (Response Amplitude Operator) for the six-degree-freedom of the hull are obtained with and without considering the tanks. Comparisons between the results with and without considering the tanks are made and significant characters regarding effects of sloshing flow on the global response of the FLNG vessel are observed.

Key words: FLNG; sloshing; coupled analysis; free surface

CLC number: U661.32    Document code: A

1 Introduction

For several decades, natural gas was merely a byproduct of oil production. Owing to its advantages, compared with other fossil fuel sources (e.g. oil and coal), the demand for natural gas is expected to increase sharply in the future. This makes the exploitation of stranded offshore gas fields attractive. However, most of the stranded gas fields locate in the sites with deep water depth and remote from the onshore infrastructure or existing offshore pipelines. As a result, FLNG, a new type of offshore unit is proposed and it has been developed as an alternative to long pipelines to onshore LNG liquefaction plant for stranded offshore fields, and research on related technology shows promising for the exploitation of the stranded gas fields[1-4].

Owing to the economical and operational reasons, FLNG is expected to be much larger than the conventional FPSO already used in oil field, in which case the sizes of the inner tank are also correspondingly large. As an offshore structure, FLNG would be located in the designated place during its service life. FLNG would be exposed to work in complex sea environments and experience different filling levels of its inner tanks. Complex sea environments together with different filling levels can cause significant sloshing problem, which would affect the
motion responses of the FLNG vessel. The tank liquid flow is altered by the vessel motion in return. This is of great concern for the FLNG operation in the production site and offloading operation of LNG carriers close to LNG terminal.

The coupling effect between ship motion and inner-tank-liquid sloshing has been studied by Molin et al [5], Malenica [6], Rognbakke and Faltinsen [7], and Newman [8] based on linear potential theory in the frequency domain. In time domain, Kim [9] studied the effect of sloshing to ship motion with 2-D sloshing calculation and Lee [10] studied sloshing effect on ship’s roll motion with 3-D calculation of single tank with 3D FDM calculation.

As mentioned above, coupled tank/ship motions have been studied by Rognbakke and Faltinsen [7], with nonlinear analyses of the interior flow in the tanks, and by Molin et al [5] and Malenica et al [6] with linear analyses. In these works, the tank dynamics are analyzed separately from the exterior radiation and diffraction problems. The solution of the coupled equations of motion is obtained by combining the hydrodynamic forces for the tanks with the vessel’s added mass, damping, and exciting forces. When the tank motions are linearized, their only effect on the vessel’s motions is to modify the added mass. Recently, a unified approach has been adopted to the panel code WAMIT to analyze the coupled tank/ship motions by Newman [8]. In this approach, the interior wetted surfaces of the tanks are included as an extension of the conventional computational domain defined by the exterior wetted surface of the body. All of the tank and hull wetted surfaces form a unified large global boundary surface. The principal modification is to impose the condition that the separate fluid domains are independent. This is achieved trivially, by setting equal to zero all coefficients of the linear system for the potential where the source and field points are in different fluid domains. This is equivalent to form separate linear equations for each domain, and concatenate these into one global system in a block-diagonal manner. The exterior free-surface Green function is used for each domain, with vertical shifts of the coordinates corresponding to the free-surface elevation in each tank. Using this approach, coupled analysis on the FLNG equipped with 10 LNG tanks is carried out.

In order to investigate the sloshing effects of the inner-tank-liquid on the global motion responses of the FLNG vessel, both numerical simulations considering the tank sloshing effects and ignoring them are conducted. Characters of the sloshing effects on the global motion responses of the FLNG vessel are observed and summed up through the comparison results between motion RAOs obtained with and without considering the tank sloshing effects.

2 Mathematical formulation

2.1 Motion formulae

The target studied a FLNG vessel equipped with 10 partially filled LNG tanks, and with zero forward speed. In the simulation, two coordinate systems such as tank-fixed and ship-fixed coordinate systems, are defined. The potential theory is adopted to describe the ship motion in waves. Decomposing the total disturbance velocity potential into an incident flow component de-
noted as $\Phi$ and a scattering potential component denoted as $\Psi$, the boundary value problem in the realm of linear theory is written as follows:

$$\nabla^2 \Phi = 0 \text{ in the fluid domain }$$

(1)

$$\frac{\partial^2 \Phi}{\partial t^2} + g \frac{\partial \Phi}{\partial Z} = 0 \text{ on the mean water surface }$$

(2)

$$\frac{\partial \Phi}{\partial n} = -\frac{\partial \Psi}{\partial n} \text{ on the body surface }$$

(3)

Radiation condition at infinite field

where $\eta$ and $n$ indicate scattering wave elevation and the normal vector on the body surface respectively.

The equation of the rigid body motions in six degree of freedom can be set up as follows:

$$[M_{ij} + a_{ij}(\omega)] \ddot{\xi} + C(\omega) \dot{\xi} + K \xi = F(\omega)$$

(4)

where $M_{ij}$ are the components of the generalized matrix for the ship hull, $a_{ij}(\omega)$ is the added mass matrix, $C(\omega)$ is wave damping matrix, and $K$ is the hydrostatic restoring stiffness matrix, and $F(\omega)$ is external force vector due to wave, mooring, sloshing, etc. In frequency domain where linear superposition rule can be applied, nonlinear term in the system needs to be linearized. For example, viscous roll damping which plays an important role in ship motion can be included by means of critical damping. And sloshing effect, even though strongly nonlinear phenomenon, can also be linearized and implemented by adding inertia and hydrostatic force into each terms in Eq.(4). The body motions corresponding to the first-order and second-order wave exciting forces can be expressed as:

$$\xi^{(1)}(\omega) = RAO(\omega) \cdot F^{(1)}(\omega)$$

(5)

$$\xi^{(2)}(\omega) = RAO(\omega) \cdot F^{(2)}(\omega)$$

(6)

where RAO($\omega$) is the Response Amplitude Operator which can be expressed as:

$$RAO(\omega) = \left[ -\omega^2 [M + a(\omega)] - i\omega C(\omega) + K \right]^{-1}$$

(7)

Once the RAO($\omega$) is obtained, response of the structure in random waves can also be obtained using linear spectrum analysis:

$$S_\xi(\omega) = |RAO(\omega)|^2 [S_f^{(1)}(\omega) + S_f^{(2)}(\omega)]$$

(8)

where $S_\xi(\omega)$ is the response spectrum of the ship motion, $S_f^{(1)}(\omega)$ and $S_f^{(2)}(\omega)$ are the first- and second-order wave force spectra, respectively.

2.2 Sloshing effect

When sloshing effect is taken into consideration in frequency domain, two facts need to be considered: inertia of sloshing fluid and restoring stiffness correction due to the inner free surface inside the tank.

The 3D panel method is also used in the calculation of the added mass of sloshing fluid. The boundary value problem for the tank sloshing can be formulated as follows:
where $\mathbf{u}$ is the velocity vector defined in the tank-fixed coordinate system. $p$, $\rho$, $\eta_{\text{inner}}$ and $\mathbf{F}$ represent the fluid density, pressure, the inner tank free-surface elevation and the external force vectors, respectively.

The presence of inner free surface causes a change of FLNG hull’s restoring stiffness. The change of restoring force due to the inclination of the ship is illustrated in Fig.1.

When the center of gravity of inner fluid $g^s$ is moved to a new position $g^s_n$ due to ship inclination of $\phi$, the whole ship’s restoring force will be decreased as much as the inner free surface’s contribution\[11\]:

$$F_{\text{Restoring}} = W \cdot \mathbf{G} M \cdot \sin\phi \cdot w^s \cdot \mathbf{g}^s\mathbf{m} \cdot \sin\phi$$

$$= (W \cdot \mathbf{G} M \cdot w^s \cdot \mathbf{g}^s\mathbf{m}) \cdot \sin\phi$$

$$= (W \cdot \mathbf{G} M \cdot w^s \cdot \mathbf{g}^s\mathbf{m}) \cdot \phi$$

$$= (W \cdot \mathbf{G} M \cdot \rho^s V^s g \cdot l^s \mathbf{g}^s\mathbf{m}) \cdot \phi$$

$$= (W \cdot \mathbf{G} M \cdot l^s \rho^s g) \cdot \phi$$

where, $w^s$ is weight of inner fluid, $\mathbf{g}^s\mathbf{m} = \frac{l^s}{V}$, $l^s$ is second moment of inertia of inner free surface with respect to $x$-axis, $V^s$ is the volume of inner fluid, $\rho^s$ is the density of inner fluid, $g$ is gravitational acceleration. The last term in the equation represents the change of restoring stiffness:

$$K^s = l^s \rho^s g$$

It can be observed that change of the restoring force due to inner fluid is affected by only second moment of inertia of inner free surface with respect to rotational axis and density of inner tank fluid, and it is not affected by filling level or location of tanks.

2.3 Coupling two problems in frequency domain

Under the assumption of small-amplitude motions of ship and liquid flow, ship motion and sloshing problems can be coupled in the frequency domain based on linearized potential flow theory. In order to take into account of the viscous effects, the linear equivalent damping co
efficient \( C_{44}^* (\omega) \) is added to \( C_{44} (\omega) \):\(^{11}\)

\[
C_{44}^* (\omega) = 2\gamma \sqrt{|M_{44} + a_{44} (\omega)| K_{44}}
\]  

(13)

where \( \gamma \) is the damping ratio of the system damping divided by critical damping. The body motion and force vectors can be written as:

\[
\zeta = R e \left( \zeta_{\text{amplitude}} e^{i \omega t} \right)
\]

(14)

\[
F(t) = R e \left( F_{\text{amplitude}} e^{i \omega t} \right)
\]

(15)

The coupling of ship motion and liquid sloshing can be investigated by adding the hydrodynamic force vectors of inner fluid motion to the right hand side of Eq.(4), and Eq.(4) is transformed into the following equation:

\[
| M_{ij} + a_{ij} (\omega) | \ddot{\zeta} + \left[ C (\omega) + C_{44}^* (\omega) \right] \dot{\zeta} + K \zeta = F_t(t) + F_s(t)
\]

(16)

where \( F_s(t) \) represents the force vector due to liquid motion. Only the inertia force of the sloshing is considered, since there is no radiation damping for the internal problem:

\[
F_s(t) = M_s\dot{\zeta} + K\zeta
\]

(17)

where \( M_s(\omega) \) is sloshing fluid’s added mass.

The hydrostatic effect of internal fluid can be included as the reduction of restoring force due to the presence of the inner free-surface.

The resulting coupled equation of motion can be written as:

\[
\left[ \left| M + a(\omega) - a'(\omega) \right| \omega^2 + i \omega \left[ C(\omega) + C_{44}^* (\omega) \right] + \left( K - K_s \right) \right] \zeta_{\text{amplitude}} = F_{\text{amplitude}}
\]

(18)

Based on the theories mentioned above, coupling analysis between ship motion and sloshing flow of inner tank liquid can be carried out. RAOs of the six-degree of freedom motions are expressed in the following section.

3 Hydro model

Before the starting of the hydrodynamic analysis, 3-D hydro models were generated. In this paper, a FLNG is chosen as the object vessel. It has a length of 392 meters, a breadth of 69.0 meters and a depth of 35.7 meters. In addition, the FLNG is also equipped with ten tanks, each of which has the same dimensions of 39m(L)×29m(B)×28m(H). It is designed to be located in the site with water depth of 1500 meters in South China Sea and moored by 12 mooring lines attached to the external turret. In the hydrodynamic simulation, a case of 26% filling level is selected with the mean draft of 13.24 meters.

The panel model on which the external wave loads are applied is presented in Fig.2. Detail of grid generation can also be observed in Fig.2. Mass model is divided into two parts: mass model of the hull shown in Fig.3 and the mass model of the inner tanks illustrated in Fig.4. Mass model of the hull which represents the mass of the FLNG hull is generated by beam ele-
ments. Correspondingly, mass model of the tanks represents the inner tank on which the sloshing effects are applied. The hydro model adopted in this simulation is the combination of panel model and mass model which is expressed in Fig.5.

The sloshing flow of inner tank liquid would become strongly nonlinear near the resonance between the sloshing flow and excitation forces. Those strong nonlinear phenomena include wave breaking, particle splash, jet flow, and impact occurrence. As we all know, it is extremely difficult to take into account all these phenomena in frequency domain due to their nonlinearities. Fortunately, these local flows seem not to play an important role in global fluid motion. In this perspective, the free-surface boundary is assumed as a single-valued function. Another nonlinear phenomenon is the viscous roll damping. It is chosen as 3% of the critical damping in the simulation.

4 Results and discussions

Based on those theories as mentioned above, motion RAOs of six-degree of freedom for the FLNG vessel are obtained. Comparison results of the motion RAOs with and without considering the tanks for the case of heading sea are shown in Fig.6. Fig.7 represents the motion RAOs with and without considering the tanks for the case of beam sea. The coupling effects between the ship motion and the sloshing flow are observed through the comparison of the motion RAOs.

Overall, it can be observed that motion RAOs of the six-degree of freedom with tanks perform the same trend as those without tanks. In this perspective, the sloshing flow of the inner tank liquid affects the ship motion in a global level. It will only affect the amplitude of the motion response, but will not change the form of motion response.
Fig. 6 RAOs of FLNG with and without tanks in heading sea
It can be concluded that the sloshing flow of the inner tank liquid can significantly affect the response amplitude of the ship motion near certain frequency, which is known as resonance frequency. And the effect becomes smaller in the areas far away from the special frequency. This significant motion response may be due to the resonant effect of the sloshing flow and the excited forces.

Among the RAOs of the six-degree of freedom, RAO for heave motion with tanks keeps almost the same as that without tanks. This indicates that the sloshing flow of the inner tank liquid has less effect on the heave motion than any other motion modes. This can be explained as follows: in a single mode of heave motion without any other motion modes, the inner tank liquid performs just as a mass block which has no sloshing effect at all. The presented minor effect on the heave motion can be expressed as one of the coupled effects of other motion modes.

Another phenomenon of great interest is that the sloshing flow of the inner tank liquid enhances the motion responses of the FLNG vessel in heading sea, while motion responses are reduced due to the effect of the sloshing flow in beam sea. This implies that the sloshing effect of inner tank liquid on the global motion of the FLNG is related to the relative angle between the vessel heading and the direction of the wave propagation. This divergent effect of the sloshing flow may be due to the different phases between the wave-excitation and sloshing-induced moments. In the case of a given filling level condition, when the wave-excitation and sloshing-induced moments have a phase difference within a range, the resultant motion can be increased due to sloshing flow. While the phase difference is out of the range, the resultant motion can be decreased owing to sloshing flow. This should be further studied and verified.

Moreover, characteristic values for the motion RAOs in both heading and beam seas are illustrated in Tab.1 and Tab.2 respectively. The characteristic values include the response frequency and its corresponding motion response. In order to provide a direct performance, column diagrams with each type motion response in a bundle were plotted in Fig.8 and Fig.9. Comparisons are made for each heading case between results with and without tanks. It is found from both Tab.1 and Tab.2 that the resonance frequency changes for the effect of the sloshing flow, compared with the case without tanks. This phenomenon is of significant use for the operation of the FLNG system in the real sea conditions. A further conclusion can be observed from the column diagrams that the resonance frequency is shifted to be smaller in sway, roll and yaw.
motions in both heading and beam seas; the resonance frequency is shifted to be larger in surge, heave and pitch motions in both heading and beam seas. Further study should be carried out on the perspective.

<table>
<thead>
<tr>
<th>Tab.1 Characteristic value for heading sea</th>
<th>Tab.2 Characteristic value for beam sea</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Motion mode</strong></td>
<td><strong>Item</strong></td>
</tr>
<tr>
<td>Surge</td>
<td>R.F.</td>
</tr>
<tr>
<td></td>
<td>C.M.R.</td>
</tr>
<tr>
<td>Sway</td>
<td>R.F.</td>
</tr>
<tr>
<td></td>
<td>C.M.R.</td>
</tr>
<tr>
<td>Heave</td>
<td>R.F.</td>
</tr>
<tr>
<td></td>
<td>C.M.R.</td>
</tr>
<tr>
<td>Roll</td>
<td>R.F.</td>
</tr>
<tr>
<td></td>
<td>C.M.R.</td>
</tr>
<tr>
<td>Pitch</td>
<td>R.F.</td>
</tr>
<tr>
<td></td>
<td>C.M.R.</td>
</tr>
<tr>
<td>Yaw</td>
<td>R.F.</td>
</tr>
<tr>
<td></td>
<td>C.M.R.</td>
</tr>
</tbody>
</table>

Note: R.F. represents Resonance Frequency; C.M.R. represents Corresponding Motion Response.

5 Conclusions

The numerical simulations have been performed by adding the 3D panel method for interior problems in frequency domain. The inner-tank sloshing effect is characterized by increase in added mass, decrease in restoring forces of sloshing fluid, and the hydrostatic correction of inner free surface. Although the frequency domain analysis is based on linear potential theory, the results generally produce the qualitative trend of the coupling effect between inner-liquid and ship motions.

Motion RAOs of the FLNG have been obtained for both heading and beam seas. Comparisons of the RAOs with and without considering the tanks are also carried out. Important phe-
nomena regarding interaction of sloshing flow and ship motion are found which can be described as follows:

1. The heave-motion amplitudes are much less affected by sloshing flow of the inner-tank liquid compared to any other motion mode in both heading and beam sea conditions.

2. In a given filling level condition, effects of tank sloshing on the global response of the FLNG can be significantly affected by wave headings. Tank sloshing can both increase and reduce the global response of the FLNG vessel due to different wave headings.

3. Resonance frequency can be shifted due to the effect of tank sloshing.

As mentioned above, the numerical simulation is based on linear potential theory. Nonlinear phenomena such as viscous damping and nonlinear sloshing flow are not taken into account in this paper. Further study on the coupled analysis between the ship motion and the sloshing flow considering the nonlinearity should be carried out. The free surface should also be further investigated.

Acknowledgments

This work was financially supported by China National Significant Scientific and Technology Major Project (Grant No. 2008ZX05026-006). This support is gratefully acknowledged by the authors.

References


频域范围内液舱晃荡对 FLNG 运动影响的研究

赵文华，胡志强，杨建民，魏跃峰
(上海交通大学海洋工程国家重点实验室，上海 200240)

摘要：FLNG 是一种新型的浮式海洋结构物，其外形与 FPSO 相似并配备天然气的液化装置以及 LNG 储罐。FLNG 概念的出现使得海上边际油气的开发不再依赖于长距离的管道运输，从而节约了此类油气田的开发成本。文中基于 WADAM 并针对液舱晃荡与船体运动之间的相互影响进行了频域范围内的研究。在研究过程中将液舱内湿表面包含于船体外湿表面中从而形成一个统一的边界表面。通过计算得出考虑液舱晃荡与忽略液舱晃荡两种情况下的六自由度运动的 RAO。对于两种情况下的计算结果进行分析研究，从而得到液舱晃荡对船体运动影响的重要规律。

关键词：大型浮式液化天然气船；晃荡；耦合分析；自由表面

中图分类号：U661.32
文献标识码：A

作者简介：赵文华(1986-),男，上海交通大学船舶海洋与建筑工程学院博士研究生；
胡志强(1976-),男，上海交通大学船舶海洋与建筑工程学院讲师；
杨建民(1958-),男，上海交通大学船舶海洋与建筑工程学院教授/博士生导师；
魏跃峰(1981-),男，上海交通大学船舶海洋与建筑工程学院博士研究生。
Performance Comparisons of Vortex-Induced Vibration Suppression Devices for Top Tensioned Riser

LI Xiao-min, GUO Hai-yan, ZHANG Yong-bo, LIU Xiao-chun (College of Engineering, Ocean University of China, Qingdao 266100, China)

Abstract: In order to reduce and even eliminate the vibrations caused by vortex shedding and increase the fatigue life of the top tensioned risers (TTR), five types of vortex-induced vibration (VIV) suppression devices were designed and the model experiments of risers equipped with suppression devices were carried out in the Physical Oceanography Laboratory of Ocean University of China. Time domain strain curves of in-line and transverse response of the model risers under the excitation of different current velocities were obtained and the corresponding model risers without suppression devices were also tested for comparison. Then the effects of the suppression devices at three different current velocities on the riser are analyzed and compared with each other. It is found that both the in-line and transverse oscillation amplitudes and frequencies of the riser with suppression devices are reduced at different degrees while their effect and efficiency on the suppression of the riser vibrations are different from each other.

Key words: top tensioned riser (TTR); vortex-induced vibration (VIV); suppression devices

CLC number: P756.2 TB535.1 Document code: A

1 Introduction

When a fluid flows about the riser, there will be shed vortices and periodic wakes which will cause the riser vibrate both in the in-line and transverse directions. When the vortex shedding frequency is locked into the riser’s natural frequency, large resonant oscillations occur and may cause serious fatigue damage. Thus the investigation of riser vortex-induced vibration (VIV) and its suppression method arouse great attention of industrial and academic circles in recent years. Reviews on the specifics of VIV in risers can be referred to Sarpkaya (2004)[1], Williamson and Govardhan (2008)[2], Gabbai and Benaroya (2005)[3].

In order to reduce and even eliminate the vibration caused by vortex shedding and increase the fatigue life of the marine riser, there are mainly two basic considerations. One is to change the characteristics of the riser to reduce the effect of vortex shedding. For example, one can change the cross section, reduced damping, top tension and structure damping of the riser. However, the above parameters can only be changed in a limited range and are normally not a
practical option. The other one is to modify the flow field by minimizing the strength of the vortices and disrupt the spanwise correlation of the vortex formation. Many suppression devices were designed such as helical strakes, fairings, alternate buoyancy joints, etc. A wide range of passive VIV suppression methods can be reviewed in Zdravkovich (1981)\textsuperscript{[4]}. During the recent years, several experiments and practical applications have been conducted to measure and study the effect of these devices. Experiments of VIV of a plain circular cylinder and cylinders fitted with helical strakes and bumps were presented in Bearman and Brankovic (2004)\textsuperscript{[5]}. Norwegian Deepwater Programme (Trim et al, 2005)\textsuperscript{[6]} commissioned experiments on riser models without VIV suppression and with various strake arrangements and current conditions and suggested that a key consideration in VIV fatigue design is the length of suppression coverage and the nature of the flow to which the bare section of the riser is exposed. Suppression fairings for vortex-induced vibration were installed in two steel catenary risers (SCRs) by Armstrong (2006)\textsuperscript{[7]}. Skaugset and Larsen (2003)\textsuperscript{[8]} dealt with suppression of VIV by introducing radial water jets from circular openings in the wall of the oscillating cylinder and a smooth cylinder with no openings was also tested for comparison. Outflow through openings in the cylinder wall is modeled through Direct Numerical Simulations (DNS) using the Spectral/hp element code, and a parameter study is performed where numbers of jets as well as jet location on the cylinder circumference and jet flow rate are varied. Owen et al (2001)\textsuperscript{[9]} carried out an experimental investigation to measure the drag and vortex induced vibration amplitudes of a circular cylinder, a circular cross-sectional body with a sinuous axis and a circular cylinder with hemispherical bumps attached. Drag reductions of about 25\% and suppression of vortex shedding have been recorded for the cylinder with bumps. Various helical strake geometries are tested by Allen et al (2004)\textsuperscript{[10]} and the effects of strake geometry, coverage length, surface roughness and interference on strake performance are discussed.

The suppression of VIV can be essential to safe production of a riser, and for many risers in deepwater today some kinds of VIV suppression devices have been installed. While there presents a key question: what types of flow improving device are presently available and which kind is more effective? The purpose of this paper is to test and compare the performances of the model risers equipped with five types of VIV suppression devices including fairings, bidirectional fairings, rectangular-tailed fairings, axial slats and 2-star helical strakes. The effect of the suppression devices and ability of the suppression devices to effectively suppress VIV are analyzed and compared with each other.

2 Experimental descriptions

2.1 Flume description

The model test was conducted in the physical oceanography laboratory of Ocean University of China. There is a two-dimensional wave generator system in the laboratory. The system includes a 65m long, 1.2m wide and 1.75m high flume, a two-dimensional wave generator, a
a wind generator, a current generator and the corresponding control system. It can be used to simulate regular and irregular wave, uniform wind speed and relatively stable current velocity. Also it can be used to combinational simulation of sea state and model test condition. Fig.1 illustrates the wind- wave- current flume.

Fig.1 Wind- wave- current flume

Fig.2 Sketch of the riser model

2.2 Experimental devices

The experimental equipments include resistance strain gauge, dynamic resistance strain instrument, dynamic data acquisition and processing system (YD-28A) type, etc.

Smooth organic glass tube is used in the model experiment. The outer diameter of the riser model is 12mm and inner diameter is 9mm. The total length of the model riser is 1.2m and the underwater length is 0.7m. The riser is vertical and the two ends are fixed on the rigid frame. The riser is allowed to oscillate in in-line and transverse directions. Three locations of the model riser are selected to place strain gauges, signed as location 1#, location 2# and location 3# as depicted in Fig.2. At each location, two strain gauges are placed as shown in Fig.3. One in the X direction is used to measure the in-line vibrations while the other one in the Y direction is used to measure the transverse vibrations.

Fig.3 Sketch of strain gauges placement

Fig.4 Suppression devices

2.3 Suppression devices

Five types of VIV suppression devices including fairings, rectangular- tailed fairings, bidirectional fairings, axial slats and 2- start helical strakes were designed and their photos and
sketches of the section are depicted in Fig.4. Fairings and helical strakes are two of the common used suppression devices in industry. The other three are new self-designed types of VIV suppression devices. The devices are all made of rubber and only equipped on the underwater part of the riser. They are represented as R1, R2, R3, R4 and R5 hereafter, respectively. The scale of the suppression devices is also depicted in Fig.4.

3 Performance comparisons of the suppression devices

The responses of all the model risers in present papers are all at location 1# with three uniform current velocities. Current velocities 0.5, 0.6 and 0.7 m/s are chosen in the analysis. The response of the riser is recorded when the vibration becomes steady. Figs. 5, 7 and 9 present the transverse response and frequency spectrum of the model risers at 0.5, 0.6 and 0.7 m/s, respectively. Figs. 6, 8 and 10 present the in-line response and frequency spectrum of the model risers at 0.5, 0.6 and 0.7 m/s, respectively. The response of the riser equipped with R4 at 0.7 m/s is not tested and the data is lack. Through comparing with each other and the bare riser, the following phenomena can be found from the figures.

R1 is more effective at suppress transverse vibration than the in-line vibration both in the strain and the frequency reduction. It can be seen from the figures that the transverse strain is suppressed by 30% and the vibration frequency is reduced by a half more compared with the bare riser. The in-line vibration strain and frequency of the riser equipped with R1 are also depressed slightly though it is not obvious as the transverse vibration.

The transverse vibration strain of the riser equipped with R2 is depressed by 90% while the vibration frequency is almost equivalent with the bare riser. The in-line vibration strain is also depressed greatly and the vibration frequency also decreases. It should be noticed that the transverse vibration frequency is equal to the in-line vibration frequency and not the general relation of 1/2.

The in-line vibration strain of the riser equipped with R3 is suppressed greatly, especially at high current velocities, and the vibration frequency is also reduced greatly. The transverse vibration strain of the riser equipped with R3 is also suppressed in some degree and the vibration frequency decreases greatly.

The in-line vibration strain of the riser equipped with R4 is suppressed greatly as same as R3 and the frequency is also reduced greatly. But different with R3, the transverse vibration strain of the riser equipped with R4 is almost equal to the bare riser though the vibration frequency decreases slightly.

The efficiency of the riser equipped with R5 on the suppression of the riser is obvious. Both the in-line and transverse vibration strain of the model riser equipped with R5 are even reduced to zero, that is to say, VIV does not happen and the model riser equipped with R5 is almost still when the current flows about the riser.
Fig. 5 The transverse response and frequency spectrum of the model risers at 0.5 m/s

Fig. 6 The in-line response and frequency spectrum of the model risers at 0.5 m/s

Fig. 7 The transverse response and frequency spectrum of the model risers at 0.6 m/s

Fig. 8 The in-line response and frequency spectrum of the model risers at 0.6 m/s
4 Conclusions and discussions

Five types of VIV suppression devices including fairings, rectangular-tailed fairings, bidirectional fairings, 2-start helical strakes, and axial slats were designed to reduce and even eliminate the vibration caused by vortex shedding and the model test was carried out in a water flume. Based on the performed comparisons of the suppression devices on the reduce of riser vibrations, the following conclusions may be drawn.

The suppression devices are found to be effective at VIV suppression and both the in-line and transverse oscillation strains and frequencies of vibrations of the riser with suppression devices are reduced at different levels. Generally speaking, fairings are more effective on the suppression of transverse vibration while bidirectional fairings and axial slats are on the in-line vibration. Rectangular-tailed fairings and 2-start helical strakes are both very effective on the suppression of transverse and in-line vortex-induced vibrations.

In general, rectangular-tailed fairings and 2-start helical strakes suppress VIV more effectively both on the suppression of transverse and in-line vortex-induced vibrations than the others in the tests. While it is also very clear that in practical engineering applications, each kind of the suppression devices may have its inherent superiority and defects. Risers equipped with rectangular-tailed fairings and bidirectional fairings may increase self-weight more greatly than risers equipped with helical strakes. Risers equipped with helical strakes and axial slats may increase drag coefficients significantly than risers equipped with fairings.
Clearly, it is imperative that a suppression system meets the technical performance criteria for a given application, while selecting the best kind of suppression devices for a given riser involves consideration of technical, installation, maintenance, and economic issues, etc. This study is only based on test data for one particular geometry of each type of suppression devices, so further research such as the type, geometry and coverage rate of suppression systems on the riser is needed.

References


Vortex-Induced Vibration Analysis of Steel Catenary Riser

RAO Zhi-biao, FU Shi-xiao, YANG Jian-min
(State Key Laboratory of Ocean Engineering, Shanghai Jiao Tong University, Shanghai 200030, China)

Abstract: The experiments of vortex induced vibrations under different current profiles are presented, which is followed by the equation of Steel Catenary Riser (SCR). Based on the energy balance principle, frequency domain theory is given to predict Vortex Induced Vibration (VIV) response of flexible riser. VIV response prediction of SCR is replaced by that of Top-Tensioned Riser (TTR). The VIV response predictions of SCR under different top-tension and different current profiles are carried out. It is found that the natural frequency varies with mode number linearly, and Root Mean Square (RMS) displacement along the riser and the number of excited mode not only depends on the magnitude and variation of current velocity, but also the top-tension.

Key words: Vortex-Induced Vibration (VIV); Steel Catenary Riser (SCR); response prediction

1 Introduction

Steel Catenary Riser (SCR) has attracted wide attention in the ocean oil industrial field. However, because of its large aspect ratio and flexible character, SCR tends to suffer from Vortex-Induced Vibration (VIV) when it is subjected to the impact of current velocity. In general, the Cross Flow (CF) displacement response of SCR under VIV is small compared to its outer diameter, but the fatigue damage due to VIV can cause extreme destruction to structural strength.

Huge efforts were put into the research activities on VIV and much progress has been made during the last few decades toward the understanding of the kinematics of VIV.

Because the current can be controlled well in indoor experiment and its cost is low compared with the outdoor experiment, many researchers focused their attentions on the indoor experimental research, trying to understand the art of VIV under different current profiles. Lie (1997) has conducted the large-scale model testing of a tensioned steel riser in shear current at Hanwytangen outside Bergen, Norway in 1997 to study if and under which circumstances the riser motions would be single-mode or multi-mode. In order to provide benchmark information
for calibration and validation of codes for predicting riser response, Trim (2005)\cite{10} conducted a VIV test with a length-to-diameter ratio of 1 400 under shear current at Marintec’s Ocean Basin in Trondheim. Chaplin (2005)\cite{11} tested a model vertical tension riser in a stepped current in a laboratory to improve the knowledges of the physics of vortex-induced vibrations of risers and to provide reliable and well-documented data to support the development of predictive models.\

The experimental researches discussed above help us understand the phenomenon of VIV, but it is far from fully understanding the art of VIV. The weakness remains wherever the model experiment has been conducted outdoor or indoor. At natural lake, the magnitude and direction of current can not be controlled conveniently, the cost of performing such experiment is also very huge. While the indoor experiment can not model the real condition of the riser in, especially the Reynolds number is not equal to the actual case. Therefore, the numerical program is often the optimal choice to predict VIV response of slender marine structures in view of experimental cost and accuracy of result. In general, empirical models were more successful at predicting cross-flow displacements and curvatures than current codes based on CFD, because two-dimensional CFD codes are incapable of adequately predicting the response envelope even for the case of vortex-induced vibrations of a stiffened cylinder on spring (Blackburn et al, 2000)\cite{12}. Among the empirically based codes, the predictions of cross-flow displacements from SHEAR 7 are more closer to measurement than other empirical code, and its prediction tends to be more conservative\cite{13}. Therefore, the empirically based code SHEAR 7 is applied to predict the cross flow VIV response of SCR.\

In this paper, the equation of SCR under the simple-support boundary condition is deduced. Then natural frequency, modal shape, displacement response and weight of participating mode of SCR under different top-tension and current profiles are given in detail. It is found that the natural frequency is proportional to square root of axis tension in ultra-deepwater. The smaller top-tension can excite higher mode than other two cases. The modal shape is not a regular sinusoid under the varying axis tension. Flow has great impact on the displacement of SCR. It is found that the smaller flow velocity can excite bigger RMS displacement than larger flow velocity.\

2 Basic theories\

2.1 Steel Catenary riser equation\

Steel catenary risers are actually flexible inclined cylinders, having initial sags and varying curvatures, which are totally different from the top-tensioned vertical risers, in view of the current direction relative to the pipe axis.\

The symbols shown in Fig.1 are explained in Tab.1.
Let the forces acting on such a SCR be split up in horizontal and vertical components in the Cartesian coordinate system, we can obtain two equations of force balance according to the static equilibrium.

In the horizontal orientation
\[ T \cdot \cos \theta - T_0 \cdot \cos \alpha = 0 \] (1)

In the vertical orientation
\[ T \cdot \sin \theta - T_0 \cdot \sin \alpha - m g s + \frac{\pi}{4} \rho_f D^2 g s = 0 \] (2)

From Eq.(1) and Eq.(2), we can obtain the differential equation
\[ \frac{d^2 z}{dx^2} = \frac{mg - \frac{\pi}{4} \rho_f D^2 g}{T_1 \cdot \cos \beta} \sqrt{1 + \left( \frac{dz}{dx} \right)^2} \] (3)

Let \( k = \frac{mg - \frac{\pi}{4} \rho_f D^2 g}{T_1 \cdot \cos \beta} \), apply the boundary condition of SCR in the lower end, namely when
\[ x=0, \ z=0 \text{ and } \frac{dz}{dx}=\tan \alpha, \text{ so } \]
\[ z=\frac{1}{k} \cosh(kx+c) - \frac{1}{k} \cosh(c) \]  \hspace{1cm} (4)

where, \( \cosh(c) = \frac{1}{\cos \beta} \cdot k \cdot H \).

And the arc length \( s \), horizontal distance \( x \) and axis tension \( T \) can be expressed by the following equations, respectively.
\[ s = \int_0^x \sqrt{1 + \left( \frac{dz}{dx} \right)^2} \, dx = \frac{1}{k} \left[ \sinh(kx+c) - \sinh(c) \right] \] \hspace{1cm} (5)

\[ x = \frac{1}{k} \ln \left| k \sinh(c) + \sqrt{1 + k^2 \sinh^2(c)} \right| - c \] \hspace{1cm} (6)

\[ T = \frac{T_0 \cdot \cos \alpha}{\cos \beta} = T_1 \cdot \cos \beta \cdot \cosh(kx+c) \] \hspace{1cm} (7)

2.2 Frequency domain theory

The amplitude response is obtained in frequency domain usually based on the principle of energy balance, namely, the energy input by the excited force into the structure equals to the energy dissipated by damping force.

The governing equation for a riser is given by
\[ m \ddot{x} + R \dot{x} - Tx'' = P(z, t) \] \hspace{1cm} (8)

where \( m \) is the mass per unit length (including the added mass). \( \ddot{x} \) is the acceleration of the structure. \( R \) is the damping per unit length (including both structural and hydrodynamic), \( \dot{x} \) is the velocity of the structure, \( T \) is the tension, \( x'' \) is the second-order derivative of the displacement of the structure with respect to the spatial variable, and the \( P(z, t) \) is the excitation force per unit length (lift force distribution).

The system displacement response can be written as the superposition of modal responses
\[ x(z, t) = \sum_{i} X_i(z) q_i(t) \] \hspace{1cm} (9)

where \( X_i(z) \) is the \( i \)th modal shape of the system. Substituting Eq.(9) into Eq.(8), have
\[ m \sum_{i} X_i(z) \ddot{q}_i(t) + R \sum_{i} X_i(z) \dot{q}_i(t) - T \sum_{i} X_i''(z) q_i(t) = P(z, t) \] \hspace{1cm} (10)

Multiply the both sides of Eq.(10) by the modal shape \( X_i(z) \), then integrate Eq.(10) over the length of riser and use the orthogonality of modal shape over the length of riser, have
\[ \ddot{q}_i(t) \int_0^L X_i^2(z) \, m \, dz + \dddot{q}_i(t) \int_0^L X_i^2(z) \, R \, dz + q_i(t) \int_0^L -TX_i''(z) X_i(z) \, dz = \int_0^L X_i(z) \, P(z, t) \, dz \] \hspace{1cm} (11)

Herein, the formula for the calculation of the \( i \)th modal force in the \( i \)th mode power-in region in the right-hand side of Eq.(11) is expressed with absolute values because it is assumed
that the local force and rth modal velocity are always in phase.

The lift force per unit length, with frequency $\omega_r$, can be written as

$$P_r(z, t) = \frac{1}{2} \rho_f DU^2 C_L(z, \omega_r) \sin(\omega_r t)$$  \hspace{1cm} (12)$$

where $\rho_f$ is the fluid volume density, $D$ is the outside diameter of riser, $U$ is the local velocity, and $C_L(z, \omega_r)$ is the lift coefficient amplitude for mode $r$. Let the modal velocity for mode $r$ be

$$q_r(t) = A_r \omega_r \sin(\omega_r t)$$  \hspace{1cm} (13)$$

where $A_r$ is the modal displacement amplitude of the structure for mode $r$.

Substitute Eqs. (12) and (13) into Eq. (11), then multiply both sides of Eq. (11) by $dq_r(t)$ and integrate Eq. (11) over the length of riser. The integrals of the first and third term on the left-hand side of Eq. (11) are zero, and the integral of the second term on the left-hand side of Eq. (11) is as follows

$$\text{Left-hand side} = \frac{1}{P} \int_0^L q_r(z) \left[ A_r \omega_r \sin(\omega_r t) \right] dz \int_0^L X_r^2(z) R(z) dz dt = \frac{1}{2} \int_0^L X_r^2(z) R(z) A_r^2 \omega_r^2 dz$$  \hspace{1cm} (14)$$

And the right-hand side of Eq. (11) becomes

$$\text{Right-hand side} = \frac{1}{P} \int_0^L q_r(z) \left[ A_r \omega_r \sin(\omega_r t) \right] dz \int_0^L X_r^2(z) R(z) \left[ \frac{1}{2} \rho_f DV^2(z) C_L(z, \omega_r) \sin(\omega_r t) \right] dz dt$$

$$= \frac{1}{2} \int_0^L X_r^2(z) \left[ \frac{1}{2} \rho_f DV^2(z) C_L(z, \omega_r) A_r \omega_r dz \right]$$  \hspace{1cm} (15)$$

In fact, Eq. (14) represents the time-average of the modal output power over one period ($P$) dissipated by damping force. And Eq. (15) stands for the time-average of the modal input power over one period ($P$) generated by excited force.

It is assumed that, for rth mode, input and output power are in balance. Therefore

$$\frac{A_r}{D} = \frac{1}{2} \int_0^L X_r^2(z) R(z) \rho_f V^2(z) C_L(z, \omega_r) dz = \frac{1}{2} \int L_1^L X_r^2(z) R(z) \rho_f V^2(z) C_L(z, \omega_r) dz$$

$$= \int_0^L X_r^2(z) R(z) \rho_f V^2(z) C_L(z, \omega_r) dz + \int L_1^L X_r^2(z) R(z) \rho_f V^2(z) C_L(z, \omega_r) dz$$  \hspace{1cm} (16)$$

where, the damping has been separated into the hydrodynamic part $C_H$ and the structural part $C_s$, because they have different integration intervals, and $L - L_1'$ denotes the length of the power-out region.

In program SHEAR 7, an initial value is assigned to the lift coefficient. An iterative calculation is then A/D performed, with the lift force and damping updating, until convergence is reached (the difference in the value between two successive calculations is within a pre-specified limit). This iterative process is completed for each mode above the cutoff. After convergence, the modal responses are used to calculate the total RMS response of the cylinder.
3 Results and discussion

3.1 VIV of SCR under different top-tension

For deepwater developments in the Gulf of Mexico, SCR's supported from both SPAR and semi-submersible platforms have proven to be a successful solution for export systems. It is envisaged that this will continue to be the most economic solution as water depths increase further, up to and beyond 10 000 feet (3048m).

The boundary condition of the SCR is pinned-pinned, the profile of current is assumed to vary with water depth, and the maximum of current is on the sea surface. Surface current velocity is chosen based on the 100-year return period hurricane data in Gulf of Mexico\cite{14}, taken as 1.28m/s. The typical export fluid densities for deepwater Gulf of Mexico developments can be 881 to 897kg/m$^3$ for oil, therefore for study work product density of 881kg/m$^3$ for oil will be assumed. Outside diameter is selected according to the suggestion of MMS (Minerals Management Service), and the SCR minimum wall thickness is selected to withstand internal design pressure during operation, hydrotest pressure, maximum allowable operating pressure and bending and external pressure. In this section, three cases are analyzed. The basic parameters of SCR are listed in Tab.2.

<table>
<thead>
<tr>
<th>Tab.2 Key data of SCR</th>
</tr>
</thead>
<tbody>
<tr>
<td>Depth of water (m)</td>
</tr>
<tr>
<td>Outside diameter (m)</td>
</tr>
<tr>
<td>Wall thinness (m)</td>
</tr>
<tr>
<td>Hang off angle (deg)</td>
</tr>
<tr>
<td>Case1 Effective tension at top (kN)</td>
</tr>
<tr>
<td>Case2 Effective tension at top (kN)</td>
</tr>
<tr>
<td>Case3 Effective tension at top (kN)</td>
</tr>
<tr>
<td>Modulus of elasticity (N/m$^2$)</td>
</tr>
<tr>
<td>Material</td>
</tr>
<tr>
<td>Density of SCR (kg/m$^3$)</td>
</tr>
<tr>
<td>Density of fluid surrounding the SCR (kg/m$^3$)</td>
</tr>
<tr>
<td>Inner fluid density of SCR (kg/m$^3$)</td>
</tr>
<tr>
<td>Uniform current velocity (m/s)</td>
</tr>
</tbody>
</table>

The VIV response of SCR in cross flow can be equivalent to corresponding VIV response of TTR. The arc length and mass per unit length of SCR are equivalent to the length and mass per unit length of TTR, respectively. Axial tension along the arc length of SCR equals that of TTR along its length. And the normal current component of SCR is the flow profile of TTR. In SHEAR7, the structural damping coefficient is 0.3%, and the Strouhal code is 200, zoneCL type is 1 (conservative force model), the still water, low and high reduced velocity region damping coefficients are 0.2, 0.18 and 0.2, respectively. Single mode bandwidth and multi-mode bandwidth are 0.5 and 0.2, respectively. The level of cutoff is 0.5.

Static displacements of SCR under different top tensions are shown in Fig.2. Slope of SCR
under the smaller top tension is smaller than that under larger top- tension at the same coor-
dinate z. In Fig.3, tensions varying with the non- dimensional arc length of SCR are displayed, and the figure also shows that the tension changes linearly approximately along the arc length except the case under the 27 300kN.

The current velocity is uniform along the water depth. The vector components of the cur-
rent normal to the axis of the SCR along the length under different top tensions are plotted in
Fig.4. The current velocity normal to the axis of SCR under larger top- tension is greater than that under smaller one. In addition, the slope of curve under the larger top tension keeps constant nearly, but the case is different to the curve under the top tension 27 300kN. Finally, all curves of velocity are fitted linearly by three lines.

The eigenfrequencies at different mode number under different top- tensions are plotted in Fig.5. From this plot, the reason why natural frequency in still water changes linearly is

\[
\frac{T/ml^2}{EI/ml^4} = 1.
\]

Therefore, the tension is dominant rather than bending stiffness, the natural frequency is proportional to mode number at low mode.

The first six modal shapes of equivalent TTR under top- tension 27 300kN along the arc
length are given in Fig.6(a). The modal shape is not strictly sinusoidal because the axis tension varies along with TTR. And the maximum value of first-order modal shape is not in the midpoint of equivalent TTR. Due to the axis tension under three top-tension varying linearly, modal shapes are similar. In order to show the modal shapes of SCR, modal shapes of equivalent TTR seen in Fig.6(a) need be transferred into modal shapes of SCR. Then, any order modal shape of any point in SCR is multiplied by 20% horizontal distance of this point. The first three modal shapes of SCR are seen in Fig.6(b), the fourth, fifth and sixth modal shapes are shown in Fig.6(c).

Fig.6 (a) The first six modal shapes of TTR

Fig.6 (b) The first three modal shapes of SCR

Fig.6 (c) The fourth, fifth and sixth modal shapes of SCR

Fig.7 shows the RMS (root mean square) value of CF displacement along the non-dimensional arc length under three top-tension. The peak of the interference pattern is clearly seen, indicating presence of the 19th, 16th and 15th mode under top-tension 27 300kN, 37 000kN and 40 000kN, respectively. Under top-tension 27 300kN, the RMS displacement fluctuates within a narrow range, because the excitation takes place in the upper region (close to surface), but downward travelling waves are damped little. However, under top-tension 37 000kN and 40 000kN, the RMS displacements fluctuates within a large range, because the riser is excited over the entire riser. The phenomenon agrees with the comment stated in Lie(2006)9.
In Fig. 8, Combinations of static and RMS displacements under three top-tension are similar to saw shape. Under top-tension 40 000 kN, seen in Fig. 8(c), the area of saw shape is larger than other two cases. And the RMS displacements in Fig. 8 are obtained by transferring the RMS displacements of TTR seen in Fig. 7 into the RMS displacements of SCR. Then RMS displacements of SCR at any point are magnified by the horizontal distance of this point.

Fig. 8 (a) Static and RMS displacements under top-tension 27 300 kN

Fig. 8 (b) Static and RMS displacements under top-tension 37 000 kN

Fig. 8 (c) Static and RMS displacements under top-tension 40 000 kN

Fig. 9 RMS value of weight with mode number

Fig. 9 shows that RMS value of mode weight factors of displacement. SHEAR 7 adopts the mode superposition method, so the weight of mode participated in can be obtained when the sinusoidal modal shapes are assumed. Modes 10-27 were selected to participate in the analysis (Lie, 2006). It is seen that modes 14-17 and 15-18 dominate the CF displacement under top-tension 40 000 kN and top-tension 37 000 kN, respectively. The wave patterns under top-tension 40 000 kN and 37 000 kN locate in the range of dominating modes. Under both top-tension 40 000 kN and 37 000 kN in Fig. 9, there is only a peak in RMS of weight, which means that the single mode locked-in happens. However, three peaks are clearly seen in RMS of modal weight under the top-tension 27 300 kN, which indicates multi-mode locked-in. The reason is that strong shear flow excites multi-mode easily. Shear flow is often defined by shear parameter (the ratio of the change in velocity of the flow over the length of the riser to the spatially averaged flow velocity over the length of the riser), see Vandiver (1996).
3.2 VIV of SCR under different current profiles

Actually, the current velocity varies with water depth. With the increase of water depth, the magnitude of current speed reduces. MMS has provided omni directional current speed at selected water depths in the Gulf of Mexico shown in Case 4 in Tab.3. The 100-year hurricane current profile has been taken from the Deepstar JIP\(^{[14]}\). In order to investigate the effect of current profile on the vortex- induced vibration response of SCR, the other two cases are chosen. The symbol ‘—’ in case 5 means that the current varies linearly along the depth of water.

### Tab.3 Current velocity

<table>
<thead>
<tr>
<th>Water depth (m)</th>
<th>Case 3 Speed (m/sec)</th>
<th>Case 4 Speed (m/sec)</th>
<th>Case 5 Speed (m/sec)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>1.28</td>
<td>1.28</td>
<td>1.28</td>
</tr>
<tr>
<td>57.9</td>
<td>1.28</td>
<td>1.28</td>
<td>—</td>
</tr>
<tr>
<td>89.79</td>
<td>1.28</td>
<td>1.28</td>
<td>—</td>
</tr>
<tr>
<td>1 822.69</td>
<td>1.28</td>
<td>0</td>
<td>—</td>
</tr>
<tr>
<td>3 048</td>
<td>1.28</td>
<td>0</td>
<td></td>
</tr>
</tbody>
</table>

The basic parameters of SCR are referenced as listed in Tab.3, and the top- tension is 37 000kN. The current velocity is constant along the depth of water in Case 3; the title Uniform current denotes this case. In case 4, the current velocity is constant initially, then it changes linearly to zero, finally it returns to constant zero, the title mix- current denotes this case. The current speed is shear in case 5; the title shear current denotes this case.

Fig.10(a) shows the variation of current velocity along the water depth. In order to predict VIV response of SCR, current velocity along the water depth should be transferred to the component of current normal to the SCR axis along the arc length. The current perpendicular to the axis of SCR along arc length and the fitting current profiles are shown in Fig.10(b).

Fig.11 shows the RMS value of CF displacement along the non- dimensional arc length under different current profiles. The peak of the interference pattern is clearly seen, indicating the presence of the 16, 17 and 15 modes under uniform current, mix- current and shear current, respectively. In case 4, no current exists in lower region (close to seabed), but the RMS displacement in lower region is the same order of that in the upper region (high velocity). The phenomenon in Case 4 in Fig.11 verifies that the excitation takes place in the upper region (close to surface), but downward travelling waves are damped little. Under uniform current,
in case 3 in Fig.11, the RMS displacement fluctuates within a large range. And there is a very interesting phenomenon that the flow velocity is smaller in the Case 4 than that in the Case 5, but RMS displacement in the Case 4 is larger than that in the Case 5.

Fig.11 The RMS of CF displacement along the non-dimensional arc length under current profile

The method of processing the RMS displacement in Fig.12 is the same as that in Fig.8. Because RMS displacements under shear current and mix current are much smaller than those under uniform current, saw shape of static and RMS displacement under shear current and mix current are not obvious. Although the RMS displacements of SCR in Fig.12(b) and Fig.12(c) are magnified by horizontal distance, combinations of static and RMS displacements under shear current and mix current are almost coincident.

Fig.12 (a) Static and RMS displacements under uniform flow

Fig.12 (b) Static and RMS displacements under mix flow

Fig.12 (c) Static and RMS displacements under shear flow

Fig.13 RMS value of weight with mode number
Fig. 13 shows RMS value of mode weight factors of displacement under different current profiles. Modes 10-22 were selected to participate in the analysis. In Fig.11, it indicates the presence of the 15, 16 and 17 modes under shear current, uniform current and mix-current, respectively. In Fig.13, it is seen that modes 15-17, 15-18 and 16-18 dominate the CF displacement under shear current, uniform current and mix current, respectively. The wave patterns under three flow profiles are located in the range of dominating modes. The maximum weight value under shear current, uniform current and mix-current is in mode number 16, 15 and 16, respectively. The RMS of weight under different current profiles increases with the mode number at the beginning, then it reaches the maximum value. Finally, it decreases with mode number.

3.3 Discussion

The current profile and characteristics in structure part are much complex for full scale long tensioned marine risers in actual ocean environment. There is loop current in the Gulf of Mexico, for example, and the internal wave has been found in the South China Sea in recent years. Simulating the profile of ocean current linearly or uniformly can not reflect the realistic current conditions. In addition, the number of response frequency of riser is more than one, which is totally different from the case of elastically mounted rigid cylindrical sections having only one eigen-frequency.

The responses of SCR under different top-tension are obtained. The smaller top-tension is, the smaller natural frequency, current profile and RMS displacement of SCR are. It is concluded that the normal component of uniform current is larger than that of shear current and that of mix-current in the same water depth. The reason why the RMS displacement of SCR under the lower current profile (mix-current) is greater than that under the higher current profile (shear current) is that the excitation taking place in the upper region (higher velocity zone) under mix-current travels to lower velocity zone.

4 Concluding remarks

In this paper, the catenary equation under the simple-support boundary condition was deducted. For SHEAR7 cannot compute VIV response of SCR with significant flexural stiffness, an equivalent vertical straight riser has been defined to replace SCR. The equivalent riser has its total length, correct tension distribution and mass distribution as the true arc length, axis tension distribution and mass distribution of the SCR, respectively. If the flow is not perpendicular to the axis of SCR, the vector component of the flow velocity will be used in the equivalent vertical straight riser.

Vortex induced vibration response predictions of SCR under three top tensions are presented. It is found that natural frequencies vary with mode number linearly. It means that the
axis tension rather than flexural stiffness is dominating the natural frequencies in ultra-deepwater. And the larger the axis tension is, the larger the natural frequency is. However, the mode shape is independent of magnitude of axis tension. The smaller top-tension can excite higher mode but smaller displacement than other two cases.

The effect of flow profiles including uniform current, shear current and mix current on the VIV response of SCR was studied. The number of excited modes under three flow profiles has little difference, but the difference of RMS displacements under three flow profiles is seen clearly. The RMS displacement under uniform current is the largest, mix current is following and the shear current is the least. In order to compare the RMS displacement with static displacement of SCR, the RMS displacement at any point has been magnified by the horizontal distance of this point. Combinations of static and RMS displacement under three flow profiles are similar to saw shape.

Acknowledgements

This work is financially supported by the National High Technology Joint Research Program of China (Grant No.2008AA09A105-6).

References

钢悬链线的涡激振动分析

饶志标, 付世晓, 杨建民

(上海交通大学海洋工程国家重点实验室, 上海 200030)

摘要: 文章介绍了立管在不同流剖面下的涡激振动实验, 然后给出了简支钢悬链线的静态方程和基于能量平衡原理的涡激振动频域预报理论。之后提出把钢悬链的参数转化为顶部张紧钻井隔水管的参数的思想。通过预报顶部张紧钻井隔水管的涡激振动响应, 得到钢悬链线的涡激振动响应。文中还研究了不同的顶部预张力和不同流剖面对钢悬链线的涡激振动响应影响。结果表明立管的固有频率和模态数的关系是线性的, 并且立管的均方根位移和激励的模态数不仅与流速的大小和变化范围有关, 还与顶部预张力有关。

关键词: 涡激振动; 钢悬链线; 响应预测

中图分类号: O427 P756.2 文献标识码: A

作者简介: 饶志标(1984-), 男, 上海交通大学硕士生; 付世晓(1976-), 男, 上海交通大学研究员; 杨建民(1958-), 男, 上海交通大学教授, 博士生导师。
Influence of Liquid Cargo in Tank on Crashworthiness of Double-skin Side Structure

CUI Jie, ZHANG A-man, YAO Xiong-liang, YANG Shu-tao
(College of Shipbuilding Engineering, Harbin Engineering University, Harbin 150001, China)

Abstract: With the successive development and application of large oil tanker and liquefied gas carrier, the damage of the ship carrying liquid cargo during collision accident is of a more and more concern. Employing the general dynamic finite element package ABAQUS EXPLICIT, the dynamic process of collision with both the case of liquid tank without oil and 80% filled is simulated. The comparisons on impact resistance, damage deformation mode, damage range and plastic deformation energy of the double-skin side structure's components including the outer shell, inner shell of the side structure and the cross framing between them are analyzed between the above two cases. It is found that the liquid cargo has little effect on outer shell, but it makes the damage range and energy absorption of the cross framing and inner shell both have a notable promotion, especially the inner shell which is directly contacted with the liquid cargo. Consequently, the liquid cargo has an extremely unfavorable influence to crashworthiness of side structure, which should be fully considered in the ship structure design.

Key words: liquid cargo in tank; double-skin side structure; crashworthiness; numerical simulation

CLC number: U661.42 Document code: A

1 Introduction

With the development of shipping industry and increasing demand for energy of the world, more and more liquid cargo ships such as Very Large Crude Carrier (VLCC), Liquefied Natural Gas (LNG) and Liquefied Petroleum Gas (LPG) tankers are developed and put into service. However, all these ships are inevitable in danger of various kinds of impact loading when they are at sea. Especially, when the side structure which is relatively weaker than other parts of the ship is struck by the other ship directly, disastrous consequences are usually caused such as environment pollution and great damage of life and property. Consequently, it is of great practical significance and value to investigate the influence of liquid cargo in tank on crashworthiness of side structure.
The interaction between the liquid cargo and the side structure under the impact loading of striking with another ship refers to the problems of transient impact, plastic flow of the material and fluid-structure interaction et al, so it is strongly nonlinear. However, previous researches were mainly focused on the effects of impact velocity, impact angle, impact mass, impact position and strain-rate sensitivity on crashworthiness of ship structure\textsuperscript{[1-4]}. Published papers with respect to influence of liquid cargo on crashworthiness of side structure are still rare so far. Anghileri et al\textsuperscript{[5]} developed numerical models for the analysis of the water sloshing in a tank during the impact with the ground and validated them using experimental data. Na et al\textsuperscript{[6]} evaluated the effect of liquid sloshing on the hull strength of a large oil tanker employing the MSC/Dytran package. In this paper, the dynamic processes of collision with both the cases of liquid tank without oil and 80% filled are simulated using the general dynamic finite element package ABAQUSEXPLICIT, and the crashworthiness of the side structure is compared between the two cases, providing some valuable references on the structural design of cargo tank and side structure.

2 Numerical models

2.1 Equations of motion on ship collision

During the collision, the equations governing the ship and liquid cargo in tank are as follows:

\[
\begin{align*}
|\mathbf{M}_s| \ddot{\mathbf{d}} + |\mathbf{C}_s| \dot{\mathbf{d}} + |\mathbf{K}_s| \mathbf{d} &= |\mathbf{F}_p| - |\mathbf{S}_{hs}|^T |\mathbf{p}| \\
|\mathbf{M}_f| \ddot{\mathbf{p}} + |\mathbf{C}_f| \dot{\mathbf{p}} + |\mathbf{K}_f| \mathbf{p} &= |\mathbf{S}_{fs}| \mathbf{F}_s
\end{align*}
\]

(1) (2)

where \(|\mathbf{M}_s|, |\mathbf{C}_s| \) and \(|\mathbf{K}_s|\) are structural mass matrix, structural damping matrix and structural stiffness matrix, respectively; \(|\mathbf{M}_f|, |\mathbf{C}_f| \) and \(|\mathbf{K}_f|\) are fluid mass matrix, fluid damping matrix and fluid stiffness matrix of the liquid cargo correspondingly; \(|\mathbf{d}| \) and \(|\mathbf{p}| \) are separately displacement of the structure and pressure of the liquid cargo; \(|\mathbf{F}_p|\) is the impact force of the striking ship and \(|\mathbf{F}_s|\) is the quantity that describes the mechanism by which the fluid drives the structure; \(|\mathbf{S}_{hs}|\) is the transformation matrix on all of the interacting fluid and structural surfaces. Combining the equations (1) and (2), the structural dynamic response and motion of the liquid cargo can be obtained finally.

2.2 Computational model and scenario of collision

Investigation results of the ship collision accidents show that great damage is usually caused when the bow of a striking ship impacts with the side structure of a struck ship. Moreover, the structural damage is mostly concentrated on the side structure of the struck ship, because the stiffness of the bow is always far higher than the side structure for a ship. Therefore, to simplify the numerical model, the striking ship is simplified as a rigid bulbous bow and the struck ship is modeled as a typical cabin block with double-skin side structure which is com-
posed of outer shell, inner shell and the cross framing (including side stringer and transverse framing) between them. In view of sway of the struck ship during collision, the coupling effect between the ship and its surrounding fluid must be taken into account. To further simplify the model, the influence of the surrounding fluid is considered as added water mass attached to the hull surface contacting with the water, which can be carried out conveniently by adjusting the density of the element of the wet surface in the numerical model. Previous research[7] showed that the added water mass coefficient could be taken as 0.4~1.3, the longer the time of collision, the larger the added water mass coefficient, and a value of 0.6 is taken in the present paper.

Assume that the rigid bulbous bow moves towards the center of the outer shell at the velocity of 6m/s and impacts vertically with the stationary struck cabin block initially, and the initial distance between the rigid striker and struck cabin is 0.01m. As for boundary conditions, the two ends of the cabin block are fixed rigidly, and the two transverse sections of the crude oil satisfy the condition that the oil can not move in the direction of y-axis. In the model, the rigid striker and struck cabin block are both described as Lagrangian parts and the crude oil is described as Eulerian part; at the same time, the techniques of single-point integration and hourglass control are taken to better dispose the nonlinear problems of large deformation and material failure. Moreover, to get a clear view of the motion of crude oil’s free surface during collision, the cabin room above the free surface is filled with Eulerian void mesh, shown as Fig.1 in detail.

2.3 Material model and failure criterion

Because the mild steel used in ship has high strain-rate sensitivity which means the yield stress and ultimate tensile strength will increase with the increment of strain rate, the effect of strain rate should be considered in the material model to investigate the dynamics of ship collision. The constitutive equation[8] by Cowper-Symonds which has a good agreement with experimental data is employed in this paper:

\[ \sigma_y/\sigma_0 = 1 + \left( \dot{\varepsilon}/D \right)^{1/2} \]  

Fig.1 Finite element model of the collision
where $\sigma_y$ and $\sigma_0$ are the dynamic yield stress and static yield stress respectively; $D$ and $q$ are Cowper-Symonds coefficients relative to strain rate, which are usually taken as 40.4 and 5 separately for the mild steel.

To be more accurately describing the material failure during collision, the ultimate equivalent plastic strain failure criterion which fits well with the Cowper-Symonds material model is used here. Based on the equivalent plastic strain of the integration point, the criterion means that when the accumulated plastic strain reaches the ultimate equivalent plastic strain of the element, it will collapse. The criterion can be expressed as:

$$\sum \left( \Delta e_p^i / e_p^i \right) = 1$$

where $e_p^i$ is the ultimate failure strain of mild steel. Considering all other influencing factors in general, a value of 0.28 is taken here.

The crude oil in tank is described with the Euler element in ABAQUS, and the material parameters in detail are as follows: the density and dynamic viscosity coefficient are 860 kg/m$^3$ and 8.8 $\times$ 10$^{-3}$ N·s/m$^2$ respectively; the bulk modulus and sound speed in oil are 2.3 GPa and 1635 m/s separately.

3 Results

To investigate the influence of liquid cargo in tank on crashworthiness of side structure, the comparisons between both the cases of tank without oil and tank filled 80% are carried out for the outer shell, inner shell and cross framing of the side structure from the aspects of impact resistance, damage deformation mode and range, and as well as energy absorption. And finally, the motion of the oil in tank is also given a preliminary research.

3.1 Impact resistance

Plots of comparison between both the cases of tank without oil and tank filled 80% are shown in Fig.2. It can be seen that the impact resistance has a strong nonlinear characteristic, and different wave crests and troughs represent various failure modes and order of different components. Generally speaking, the trend of them shows an overall agreement, while the liquid cargo makes the fluctuation of the impact resistance to be gentler and the corresponding phase of wave come earlier than the case of tank without oil. To be specific, they show the best agreement during the initial 10ms when the impact resistance increases sharply with time increment until the peak value is reached, and then it follows a sharp decrement, which indicates the deformation and fracture of cross framing of the side structure; subsequently, the deformation of the outer shell reaches its limits with the penetration of the striker and ruptures around the time of 19ms; finally the resistance comes into the stage of vibration attenuation, resulting from the combined action of inner shell and motion of the oil in tank.
Moreover, it can be seen that the side structure has the same the failure order in the two cases viz. crossing framing, outer shell and then inner shell. However, the influence of the liquid cargo makes the fracture moments of the side structure’s components come earlier with different degree, especially the fracture moment of inner shell which is considered as an important symbol of liquid cargo tanker’s safety is changed from 38ms to 27ms after collision, which is great harmful to crashworthiness of the side structure.

3.2 Damage and energy absorption of side structure’s components

In order to further study the effect of liquid cargo on crashworthiness of side structure, the damage deformation and energy absorption of outer shell, inner shell and cross framing with both the cases of tank without oil and tank filled 80% are plotted in Figs.3- 8.

Fig.3 and Fig.4 are respectively the plots of the damage deformation and energy absorption of the outer shell. As can be seen, in both of the cases the deformation modes mainly include membrane tensile, plate bending and fracture, and the range of deformation is nearly the same, with the size of fracture totally depending on the diameter of the striker struck into the
side structure; the curves of the energy absorbed by the outer shell have a great agreement with each other, and the energy absorption is relatively larger than the case of tank without oil, with a maximum increase of 4.5%. So the liquid cargo in tank has little effect on crashworthiness of the outer shell.

Fig.5 and Fig.6 are the energy absorption and damage deformation plots of the cross framing. Compared with the damage deformation modes of buckling, crushing and shearing splitting in the case of tank without oil, the cross framing forms one more deformation mode viz. outward large plastic deformation in the case of tank filled 80%, which is mainly caused by the sloshing of oil in the tank; at the same time, the damage deformation range of the cross framing is also extended from the striker’s directly contact zone to the area of plate panels supported by the strong framework outside the edge of rigid striker. From the aspect of energy absorption, the curves of energy absorbed by the cross framing in the two cases are nearly the same or coincide with each other at initial 12ms, which shows that the effect of the oil in tank has not been fully taken on; subsequently, the energy absorption is promoted rapidly in the case of tank filled 80% with a maximum increase by 55% compared with the energy absorption in the case of tank without oil, which is caused by the combined action of the rigid striker and oil in tank. So the liquid cargo has a notable impact on crashworthiness of the cross framing.
The contour plot of the inner shell's damage and its energy absorption curves are shown in Fig.7 and Fig.8. As can be seen from Fig.7, the damage deformation modes of the inner shell are totally the same as the outer shell in the case of tank without oil while one more new mode namely outward large plastic deformation which is supported by the strong cross framing appears on the inner shell in the case of tank filled 80%; at the same time, the deformation rang has almost been extended into all over the inner shell. In Fig.8, it is found that the change trend of energy absorbed by the inner shell is alike to the cross framing which is discussed above and the energy absorption in the case of tank filled 80% is 15 times than that in the case of tank without oil, which shows that the crashworthiness of the inner shell can be significantly affected by the motion of liquid cargo.

Fig.7 Comparison of the inner shell’s damage

Fig.8 Energy absorbed by the inner shell

Fig.9 Energy absorbed by the whole side structure

To give a comprehensive analysis on the influence of liquid cargo on crashworthiness of double-skin side structure, the energy absorption of the whole structure is plotted in Fig.9. During the initial 12ms after collision, the time history of energy absorbed by the side structure in the two cases nearly coincides with each other; then the impacts of liquid cargo start to take effect, and the energy absorption reaches 1 611KJ finally under the coupling effect between the oil and the inner shell, compared with the energy absorption 534KJ in the case of tank without oil. Consequently, the liquid cargo in tank has an extremely unfavorable effect on crashworthiness of side structure, which should be fully taken into account in the process of structural design.
It is worth to be concerned that the deformation of the inner shell located on the other side of shipboard is also very notable in the case of tank filled 80%, shown as Fig.10. On the whole, the entire inner shell has different degree of inward plastic deformation supported by the strong cross framing, with the maximum value appears near the free surface of the oil. The reasons which can explain the above phenomenon are that the oil absorbs some energy during collision and then slams on the inner shell located on the other side in the form of wave sloshing. Especially when the first wave reflection happens, the inner shell deforms greatly.

3.3 Motion of the liquid free surface

To get a clear view of the motion of liquid free surface under the impact resistance loading, the middle section profiles of the oil at various times are plotted in Fig.11. Generally, the motion of the oil free surface can be divided into 3 stages: forming of the wave, developing of the wave and closing of the wave. During the stage of forming of the wave, only the free surface contacts or near the inner shell rises a little and has a trend of moving forward with a relatively lower wave peak, shown as Fig.11(a)~(c); at the stage of developing of wave, the free surface lifts further and moves towards the other side of the tank until the wave surface slams on the upper deck or the inner shell in the other side with the wave surface breaking, as shown in Fig.11(d)~(f); finally, it comes the stage of wave closing. During this stage, the liquid becomes to backflow under the reflection of all plating of the liquid tank and starts to close under the action of gravity, forming local cavity at some instants, shown as Fig.11(g)~(i).
4 Conclusions

The dynamic process of ship collision in both the cases of tank without oil and tank filled 80% is simulated in the current paper, and the main conclusions are as follows:

1. From the views of impact resistance, the liquid cargo makes the fluctuation of the impact resistance to be gentler and the corresponding phase of wave comes earlier than the case of tank without oil.

2. Under the influence of liquid cargo, the fracture moments of the side structure’s components come earlier with different degree, especially the fracture moment of inner shell which is considered as an important symbol of liquid cargo tanker’s safety is changed from 38ms to 27ms after collision, which is great harmful to crashworthiness of side structure.

3. From both the aspects of damage deformation mode and range, the effect of liquid cargo on the outer shell is very little and can be neglected to some extent; while for the cross framing the range of damage deformation is extended obviously; as for the inner shell, one more damage deformation mode namely outward large plastic deformation appears compared with the case of tank without oil and the range of damage has almost extended all over the inner shell. Consequently, the liquid cargo has a significant influence on crashworthiness of side structure.

4. With respect to energy absorption, the energy absorbed by cross framing is increased by 55% and the energy absorbed by the inner shell and the whole side structure in the case of tank filled 80% are respectively about 15 and 3 times than that in the case of tank without oil, besides the little increase of energy absorption of the outer shell.

5. According to the dynamic characteristics of liquid cargo during collision, the motion of the liquid free surface in tank can be divided into 3 stages: forming of the wave, developing of the wave and closing of the wave.

References

船舶内液货对双层舷侧碰撞性能的影响分析

崔杰, 张阿漫, 姚熊亮, 杨树涛
(哈尔滨工程大学船舶工程学院, 哈尔滨 150001)

摘要: 随着大型油轮及液化气船的相继开发和应用, 舱内液货在碰撞中的损伤问题越来越备受关注。采用大型动力非线性有限元软件 ABAQUSEXPLICIT, 对空载和 80% 载荷两种状态下的碰撞动力学过程进行了数值仿真计算。通过两种装裁状态下碰撞力载荷、舷侧各构件(舷侧外板、内板及之间十字隔板)损伤变形模式、范围及塑性变形能吸收等的对比分析发现: 舱内液货对舷侧外板影响不大, 但十字隔板及舷侧内板在损伤范围和能量吸收上均得到了很大的提升, 尤其以与液货接触的舷侧内板最为显著。这说明舱内液货对船舶舷侧碰撞性能影响极为不利, 在结构设计时必须予以充分考虑。

关键词: 舱内液货; 双层舷侧; 碰撞性能; 数值仿真

中图分类号: U661.42
文献标识码: A

Comparative Studies of the Transverse Structure Design Wave Loads for a Trimaran by Model Tests and Rule Calculations

WANG Xue-liang, HU Jia-jun, GU Xue-kang, GENG Yan-chao, XU Chun
(China Ship Scientific Research Center, Wuxi 214082, China)

Abstract: It is very important to obtain the cross structure design wave loads of a trimaran for its transverse strength in structure design. In order to study the cross structure design wave loads of a trimaran, its model test was performed in a seakeeping tank. The model test conditions were composed of regular and irregular waves in head and quartering seas. Splitting moments $M_{sp}$, splitting shear forces $Q_{sp}$ and transverse torsional moments $M_{tt}$ of the cross structure were measured in model tests. RAOs and statistical properties were analyzed in detail. According to analysis of the cross structure wave loads from model tests, deductions and rule calculations, the full amplitude wave height to the calculation of those transverse wave loads might be 6.0 meters in LR rule. To different transverse wave loads, the extrapolated result of design wave loads with dependability 99% of transverse torsional moments $M_{tt}$ was greater than that from rule calculation. However, the deductions of splitting moments $M_{sp}$ and splitting shear forces $Q_{sp}$ accorded with rule calculations. It can be concluded that the evaluation methods from LR rules are some nonconservative for the cross structure design of transverse torsional moments $M_{tt}$. This should be paid some attentions to ship designers in particular.

Key words: trimaran; cross structure; design wave load; rule; model test

1 Introduction

Different from traditional monohull ship, trimaran will be exposed to transverse wave loads except for longitudinal wave loads when traveling at sea because of the existence of cross structure of this kind of ship. These transverse wave loads include splitting moments $M_{sp}$, splitting shear forces $Q_{sp}$ and transverse torsional moments $M_{tt}$. There will have interaction from water between side hulls and main hull. Furthermore, the fluid force acting on a side hull will transfer to the main hull by cross structure. Then a trimaran will be exposed to different wave loads comparing to those of monohull ship when traveling at sea.

The measurement of wave loads of a trimaran should be given key consideration of the particularity of cross structure because of the different type of wave loads. The proper measurement method of transverse wave loads should be studied to $M_{sp}$, $Q_{sp}$ and $M_{tt}$. And the wave loads measuring system should be designed to match this method. Wang (2009) had given a...
further study on the measuring method of a trimaran wave loads.

John Hampshire et al (2004) had taken a wave loads model test of a RV TRITON trimaran in the Ocean Basin at QinetiQ Haslar (UK). The model was divided into five segments along its length of the main hull and two segments in each side hull. A beam of varying stiffness connects both the main hull and side hull segments. The side hulls were attached to the main hull by two beams, one for each segment. The transverse wave loads which had been measured in model tests were vertical moments, vertical shear forces, horizontal moments, horizontal shear forces and torsional moments.

Kennell (2004) had taken a model test of a high speed strategic sealift trimaran in David Taylor Model Basin (USA). The main hull of the model was converted into a six-segment hydrodynamic loads model using a calibrated beam or back-spline that linked together the segmented main hull. Segment breaks were at stations 4, 7, 10, 13, and 16. Stations 4 and 16 were instrumented to measure vertical and lateral bending moments, torsional moments, and vertical and lateral shear forces. Stations 7, 10, and 13 were instrumented to measure vertical and lateral bending moments only. Side hulls were connected to the cross structure through load cells to allow measurement of side hull bending, shear, and torsion.

Lloyd’s Register had its corresponding design wave loads rules for the classification of trimarans. The splitting moments, splitting shear forces and transverse torsional moments were given as provisions to cross structure of trimarans.

Dai et al (2007) had pointed out that the ship design wave loads were determined by three principles: rationality, uniqueness and simplicity. Thus these three principles can be used to judge which value was suitable when ship designers use LR rules or model tests to determine the design wave loads. LR rules were used to evaluate the design wave loads of cross structure for a trimaran in this paper. And the evaluating values were compared and analyzed with model tests and their extrapolated values from Ochi method. Then the applicability of LR rules to evaluate the design wave loads of cross structure for this trimaran was clear. The study result had certain significance to structural design of the same type trimaran.

2 Model tests and results analysis of a trimaran for its cross structure

This kind of trimaran had a high cruising speed and two side hulls located in post median of the main hull. The cross structure connected the main hull and two side hulls. The characteristic of this ship form made trimarans had a good seakeeping performance. The preconcerted service area of this trimaran had the wave height for cumulative distribution which was shown in Fig.1 (introduced from reference [7]). It can be seen from this figure that waves took 71 per-

![Fig.1 Wave height for cumulative distribution of preconcerted service sea area](image-url)
cents within 2.5 meters height, 91 percents within 4.0 meters height and 98.5 percents within 6.0 meters height of all waves.

2.1 Model feature

The test model of this trimaran was made by fiber reinforced plastics (FRP), and its main parameters were shown in Tab.1. In order to measure the longitudinal and transverse wave loads in different traveling status, segment breaks were at stations 4, 8.5 and 13, see Fig.2. Beam stiffness at each segment break corresponded to that of the main hull. Thus the testing beam was a varying stiffness one along the main hull. Each side hull was cut into two segments at station 4 and was connected by a short beam. On the other hand, each side hull was attached to the main hull by a uniform beam which located at station 4.5. The height directions of these two beams were located in the center of gravity of each side hull. The measurement contents, methods and the locations in detail were presented in Tab.2. Thus, seven beams totally constituted the whole measuring frame of wave loads.

Tab.1 Main parameters of a trimaran model

<table>
<thead>
<tr>
<th>Main hull</th>
<th>Side hull</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length (m)</td>
<td>3.843</td>
</tr>
<tr>
<td>Waterline length (m)</td>
<td>3.684</td>
</tr>
<tr>
<td>Breadth (m)</td>
<td>0.295</td>
</tr>
<tr>
<td>Depth (m)</td>
<td>0.316</td>
</tr>
<tr>
<td>Draft (m)</td>
<td>0.150</td>
</tr>
<tr>
<td>Displacement (kg)</td>
<td>86.8</td>
</tr>
</tbody>
</table>

2.2 Model tests and data analyses

In order to give the comparison of model results and rule calculations, the testing states with speed 2.124m/s, wave height 131.6mm and heading sea 135°, 90° and 45° in regular waves were performed. RAOs of Msp, Qsp and Mtt in these states were shown in figures 3 to 5. And the results in irregular waves would be analyzed in next section’s comparisons.

From Fig.3 it can be seen that Msp was bigger in beam wave than that in oblique wave. However, it was difficult to obtain the Msp values in shorter wave lengths because of the limitation of wave making ability. The peak value of Msp in heading oblique waves was greater than that in following oblique waves, and the peak of responses located at the ratio of wave length to ship length being about 0.9.

It can be seen from Fig.4 that Qsp had the similar current to Msp for their relationships to wave direction. But the peak value in oblique waves moved to the direction to big ratio of wave length to ship length and some hypo-peak values appeared.

Fig.5 showed the RAOs of transverse torsional moments Mtt. It can be seen that the peak of RAOs curves had the highest value in following oblique waves, and there had the middle peak value in heading oblique waves, and the smallest peak value appeared in beam wave. Similar-
ly, it was also difficult to obtain the $M_{tt}$ values in shorter wave lengths because of the limitation of wave making ability.

3 Extrapolated design loads

According to Ochi methods, if the main short term statistics of wave loads had been obtained and their peaks obeyed the Rayleigh distribution, the short term extreme value $\gamma_n$ of loads was given by the following formula.

$$\gamma_n = \sqrt{2\ln\frac{n}{\alpha}} \sqrt{m_0}$$

where, $\alpha$ = probability that short term extreme value exceeds a specified value; $n$ = number of observations; $m_0$ = area under spectral density function of loads responses.

It was generally taking $\alpha$ as 0.01 to global design loads. The extrapolated results denoted the likely encountered maximum of loads under a wind wave with 99 percents probability guarantee.

4 Rules calculation of design wave loads

LR rules for trimarans had defined the splitting moments $M_{sp}$ which was showed in Fig.6. $M_{sp}$ and $M_{sp}$ were according to moments of hogging and sagging respectively. And the corresponding locations were points I and O. In order to have the right comparison with model tests values, point I was chosen to calculate the splitting moments $M_{sp}$ in this paper. Furthermore, restrictions such as cruising speed and service sea area should be concluded in rules calculations.

5 Comparison analyses of design wave loads

The comparison among cross structure wave loads from mod-

Fig.5 RAOs of transverse torsional moments

Fig.6 The definition of splitting moments $M_{sp}$

Fig.3 RAOs of splitting moments

Fig.4 RAOs of splitting shear forces
el tests, extrapolation and LR rules were given in Tab. 2. Here the sea states were heading oblique wave with the ratio of wave length to ship length 0.9, 1.0 and 1.1, and wave height had different full amplitudes 2.5m, 4.0m and 6.0m, and the speed was 2.124m/s, and the wave direction was 135°. Meanwhile, the model tests results and the extrapolated results corresponding to the significant wave height 2.5m and 4.0m in irregular waves were also given in Tab. 2.

### Tab. 2 Comparison among cross structure wave loads from model tests, extrapolation and LR rules

<table>
<thead>
<tr>
<th>Load type</th>
<th>Wave height</th>
<th>Regular waves (Full amplitude)</th>
<th>Irregular waves</th>
<th>Extrapolated results</th>
<th>Rules calculations</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>( \lambda/L = 0.9 )</td>
<td>( \lambda/L = 1.0 )</td>
<td>( \lambda/L = 1.1 )</td>
<td>Hogging</td>
</tr>
<tr>
<td>Msp (kNm)</td>
<td>2.5m</td>
<td>2710.3</td>
<td>2083.9</td>
<td>1738.7</td>
<td>562.5</td>
</tr>
<tr>
<td></td>
<td>4.0m</td>
<td>2978.8</td>
<td>2991.6</td>
<td>2556.9</td>
<td>818.2</td>
</tr>
<tr>
<td></td>
<td>6.0m</td>
<td>3029.9</td>
<td>4091.0</td>
<td>3707.5</td>
<td>—</td>
</tr>
<tr>
<td>Qsp (kN)</td>
<td>2.5m</td>
<td>405.0</td>
<td>366.0</td>
<td>479.8</td>
<td>176.3</td>
</tr>
<tr>
<td></td>
<td>4.0m</td>
<td>469.7</td>
<td>471.0</td>
<td>497.9</td>
<td>360.0</td>
</tr>
<tr>
<td></td>
<td>6.0m</td>
<td>886.8</td>
<td>839.7</td>
<td>792.0</td>
<td>—</td>
</tr>
<tr>
<td>Mtt (kNm)</td>
<td>2.5m</td>
<td>3400.7</td>
<td>3247.3</td>
<td>3823.5</td>
<td>4103.8</td>
</tr>
<tr>
<td></td>
<td>4.0m</td>
<td>8194.9</td>
<td>8246.0</td>
<td>7287.2</td>
<td>16764.4</td>
</tr>
<tr>
<td></td>
<td>6.0m</td>
<td>10278.7</td>
<td>9848.7</td>
<td>10125.3</td>
<td>—</td>
</tr>
</tbody>
</table>

From Tab. 2 it can be seen that transverse wave loads of cross structure had a nonlinear increasing current with the wave height increment. As to the three regular wave lengths (\( \lambda/L = 0.9, 1.0 \) and 1.1), the nonlinear phenomena were not so much clear, and the reason might be the rather short distance among these three wave lengths. Msp in state with \( \lambda/L = 1.0 \) and wave height 6.0m corresponded with the rules calculation. Qsp in state with \( \lambda/L = 0.9 \) and wave height 6.0m corresponded with the rules calculation. Mtt in state with wave heights 2.5m to 4.0m corresponded with the rules calculation. On the other hand, the model tests results in irregular waves had an inapparent asymmetry of hogging and sagging parts of moments. And this might be caused by the lack of simulation of the whole wet deck of this trimaran, and thus it would result in the difference of hogging and sagging parts of cross structure wave loads became small. Rule calculations of Msp and Qsp were bigger than those from model tests. This showed that the corresponding significant wave heights to rule calculations of Msp and Qsp should have a bigger value than 4.0m. Comparing the extrapolated results to model tests results in irregular waves, the extrapolated results of Msp and Qsp corresponded to those from rule calculations with significant wave height 4.0m and 2.5m to 4.0m, respectively. However, the extrapolated results of Mtt was greater than those from rule calculations with significant wave height 2.5m.

### 6 Results

Study of the cross structure design wave loads for a trimaran in regular and irregular waves
using model testing method is presented in this paper. And the extrapolated results of these transverse wave loads were given using some model testing results in irregular wave. Evaluation of these transverse wave loads of a trimaran using LR rules was also performed in this paper. From the comparison among the above model testing results, extrapolated results and rule calculations, some valuable results were obtained:

(1) \( M_{sp} \) was bigger in beam wave than that in oblique wave. However, it was difficult to obtain the \( M_{sp} \) values in shorter wave lengths because of the limitation of wave making ability. The peak value of \( M_{sp} \) in heading oblique waves was greater than that in following oblique waves, and the peak of responses located at the ratio of wave length to ship length being about 0.9.

(2) The transverse wave loads of cross structure had a notable nonlinearity to wave heights. These transverse wave loads from LR rule calculations might correspond to the regular wave height (full amplitude) 6.0m.

(3) The inapparent asymmetry of hogging and sagging parts of moment from rule calculations was much more notable than that from model test results. It might be caused by more considerations of nonlinear features of these cross structure wave loads.

(4) For different type of transverse wave loads, the extrapolated value with 99 percent probability guarantee to transverse torsional \( M_{tt} \) was greater than that from rule calculations. However, the extrapolated value of splitting moments \( M_{sp} \) and splitting shear force \( Q_{sp} \) were corresponding to that from rule calculations. It indicates that the evaluation methods from LR rules are nonconservative for the cross structure design of transverse torsional moments \( M_{tt} \).

References

三体船横向结构波浪设计载荷试验与规范比较研究

汪雪良，胡嘉骏，顾学康，耿彦超，徐春
（中国船舶科学研究中心，江苏无锡 214082）

摘要：三体船连接桥结构波浪设计载荷的确定对结构设计中关注的横向强度问题来说是非常重要的。为了研究一艘三体船连接桥横向波浪载荷，进行了该三体船的水池模型试验，包含不同波向下的规则波与不规则波试验。试验对连接桥遭受的分离弯矩Msp，分离剪力Qsp和横向扭矩Mtt进行了测量，分析了其传递函数和统计的特性。根据对这些连接桥横向波浪载荷在试验、推断值与规范计算值之间的比较分析表明：劳氏规范关于这些横向波浪载荷的计算值可能对应于规则波双幅波高 6.0m；对于不同的横向波浪载荷来说，保证率为 99% 的设计载荷推断值对横向扭矩 Mtt 来说要大于规范计算值，而分离弯矩Msp和分离剪力Qsp的推断值对应的规范计算值则相当，这意味着设计人员在用劳氏规范进行校核时，横向扭矩Mtt 的规范计算值是偏向于冒险的，应给予特别关注。

关键词：三体船；连接桥；波浪设计载荷；规范；模型试验

中图分类号：U661.72 文献标识码：A

作者简介：汪雪良（1977-），男，中国船舶科学研究中心高级工程师，博士研究生，研究方向为船舶与海洋工程波浪载荷、结构响应与试验研究等，E-mail: wsnowolf@126.com。
胡嘉骏（1965-），男，中国船舶科学研究中心研究员。
顾学康（1963-），男，中国船舶科学研究中心研究员。
耿彦超（1982-），男，中国船舶科学研究中心助理工程师。
徐春（1980-），男，中国船舶科学研究中心工程师。
A Comparison of Different Rules for the Spherical Pressure Hull of Deep Manned Submersibles

PAN Bin-bin, CUI Wei-cheng
(China Ship Scientific Research Center, Wuxi 214082, China)

Abstract: A brief description of current available design rules for spherical pressure hull of deep manned submersibles is provided, and the common foundation of these rules is found to be either the classical yielding load formula or critical elastic buckling load formula. Then these rules are used to calculate the shell thickness of the spherical pressure hulls of several existing submersibles. The calculation results are compared after the unification of input parameters of all design rules. According to this comparison, it is found that there is significant difference among the calculation results of these rules and the design thickness calculated by most rules is much larger than the actual design thickness of existing pressure hulls. Thus it is appealed to be urgent and necessary to update and unify current design rules. Finally, the process of establishing the new design rule is proposed.

Key words: spherical pressure hull; manned submersible; design rule; safety

CLC number: U662.1  Document code: A

1 Introduction

Oceans cover more than 70% of the earth, but most of them are unexploited. It is true that oceans are still shrouded in mystery, especially the dark deep-sea. As an important deep-sea survey tool, a deep manned submersible provides a safe operation space for human beings in deep-sea investigation and development. The most critical component of a deep manned submersible is the manned pressure hull. It provides not only a safe living space for pilots and scientists but also provides a proper working condition for non-pressure-resisting and non-water-repellent equipments. So the pressure hull should be designed to have enough strength and water-tightness. At the same time, the weight of the pressure hull should be as light as possible since it occupies a large part of the weight of the submersible. As the most commonly used pressure hull type, spherical pressure hull has been used in all the existing deep manned submersibles such as Alvin of USA, Nautile of France, RUS (Consul) of Russia, Shinkai6500 of Japan and Jiaolong of China[1]. The strength and stability of spherical pressure hulls have been studied since 1915[2], and many theoretical, experimental, and numerical investigations have been carried out about the load carrying capability of spherical shells, e.g., Yokota et al[3]; Fan[4]; Paliy[5]. A recent overview of buckling and ultimate strength of spherical pressure hull
under external pressure has also been conducted\[^{6}\]. Some research results have been used to produce design rules. In order to design a new 4 500m deep manned submersible, the current available design rules from ABS\[^{7}\], BV\[^{8}\], CCS\[^{9}\], DNV\[^{10}\], GL\[^{11}\], LR\[^{12}\], NK\[^{13}\] and RS\[^{14}\] are collected and compared in this paper. According to this comparison, significantly different results are found. Furthermore, calculations of the spherical pressure hulls of existing deep manned submersibles mentioned above are carried out. These calculations show that many existing spherical pressure hulls are not in compliance with most current design rules. But the practical experiences of above existing manned submersibles have not been reported to meet any safety problems related to the load carrying capacity of spherical hulls. Both above significant differences among current design rules and the lack of agreement between actual hulls and the calculation of design rules indicate that current design rules need to be updated and unified like Common Structural Rules for tankers and bulk carriers\[^{15}\]. The primary purpose of this paper is to present the comparison of current design rules and the check of fulfillment of pressure hulls of existing manned submersibles to these design rules. In addition, it also discusses the common basis of current design rules and the process of establishing a unified design rule. 

2 Description of current design rules

Current available design rules of manned submersibles which are compared in this paper are from ABS\[^{7}\], BV\[^{8}\], CCS\[^{9}\], DNV\[^{10}\], GL\[^{11}\], LR\[^{12}\] and RS\[^{14}\]. Due to the unavailability of the design equations of NK design rules\[^{13}\], it is not used in this comparison. The calculation methods of spherical pressure hull of these design rules are described briefly in this section. The common parts of these rules are also summarized during this description. For the sake of clarity and simplicity, the names of design rules are represented by abbreviation, for example, rules for certification/classification of submersibles which were published by Det Norske Veritas (DNV) in 1988 are abbreviated to DNV1988.

2.1 DNV1988

The primary calculation equations of spherical hulls of DNV1988\[^{10}\] are Eq.(1) and Eq.(2).

\[ P \leq \frac{P_{cr} \cdot \psi}{\gamma \cdot \gamma_m \cdot \kappa} \]  
\[ P_{cr} = 2 \cdot \Phi \cdot \frac{t}{R_m} \cdot \sigma_y \]  

where \( \psi, \gamma, \gamma_m, \kappa \) are coefficients reflecting post buckling behavior, load factor (safety factor), material factor and structural slenderness factor, respectively. All these coefficients can be treated as the additional factors which represent the uncertainty of structure and material. \( \Phi \) is the plasticity modification factor which reflects the slenderness and imperfection of structure, as shown in Eq.(3) to Eq.(6).

\[ \Phi = \frac{1}{\sqrt{1 + \lambda^2}} \]
\[ \lambda = \sqrt{\frac{\sigma_y}{\sigma_e}} \quad \text{(4)} \]
\[ \sigma_e = 0.605 \cdot \delta \cdot \frac{t}{R_m} \cdot E \quad \text{(5)} \]
\[ \delta = \frac{0.5}{{\sqrt{1 + \frac{R}{100 \cdot t}}} } \quad \text{(6)} \]

According to Eq.(2), it can be found that the calculation of DNV 1988 is mainly based on the classical yielding load which is shown in Eq.(7). The effects of elastic buckling which is given in Eq.(8) and structural imperfection are compiled into a plasticity modification factor \( \Phi \) (Eq.(3))

\[ P_y = \frac{2\sigma_y \cdot t}{R_m} \quad \text{(7)} \]
\[ P_e = \frac{2E}{ \sqrt{3(1 - \nu^2)} } \cdot \left( \frac{t}{R_m} \right)^2 \quad \text{(8)} \]

Actually, when \( \nu = 0.3 \), Eq.(5) can be rewritten as:

\[ \sigma_e = \delta \cdot \frac{P_e \cdot R_m}{2 \cdot t} \quad \text{(9)} \]

And Eq.(4) can be rewritten as:

\[ \lambda = \sqrt{\frac{P_y}{\delta P_e}} \quad \text{(10)} \]

Here, we can find that the classical yielding load \( P_y \) and critical elastic buckling load \( P_e \) play an important role in the calculation equations of DNV1988.

2.2 BV1989

BV1989\(^{[8]}\) treats safety factor \( sf \) as a function of structural imperfection \( w \), as shown in Fig.1. The cubic function obtained by curve fitting to the original curve is also given in Fig.1.

![Fig.1 Curve fitting of BV1989\(^{[8]}\)](image)

\[ P \leq P_y / sf \quad \text{(11)} \]
\[ P_y = \min(P_{\text{el}}, P_f) \quad \text{(12)} \]

In Eq.(12), \( P_{\text{el}} \) is the failure pressure due to elastic instability, as given in Eq.(13). Compared to
Eq. (8), it can be regarded as a modified critical elastic buckling load. $P_i$ is the failure pressure due to membrane yield, as shown in Fig. 1, where the sixth polynomial function is also given.

$$P_{el} = \frac{9.6E}{9 + 0.003\left(\frac{2R_o}{t}\right)^2}$$  \hspace{1cm} (13)

In Fig. 1:

$$P_p = \frac{2t\sigma_y}{R_o}$$  \hspace{1cm} (14)

$$P_m = \frac{2E}{\sqrt{3(1-\nu^2)}}\left(\frac{t}{R_o}\right)^2$$  \hspace{1cm} (15)

It can be found that the difference between Eq. (14) and Eq. (7) is $R_o$ and $R_m$, the same difference exists between Eq. (15) and Eq. (8). So $P_p$ can be regarded as the modified yielding load and $P_m$ can be regarded as critical elastic buckling load.

2.3 LR1989

In the rule of LR1989\cite{12}, the operating pressure is expressed as the function of critical elastic buckling load ($P_e$) and the modified yielding load ($P_s$), as shown in Fig. 2.

$$P_i = \gamma_m \cdot \frac{2\sigma_y t}{R_m}$$  \hspace{1cm} (16)

where $\gamma_m$ is the material factor and 1.5 can be treated as the safety factor.

2.4 CCS1996

The calculation equation of CCS1996\cite{9} is separated into two parts. The first part is failure pressure due to membrane yield, as shown in Eq. (17). The second part is due to buckling, as shown in Eq. (18).

$$P = \frac{1}{1.765}P_y$$

In Eq. (17), 1.765 can be regarded as the safety factor.

$$P \leq P_{cr} = C_z C_s P_{el}$$  \hspace{1cm} (18)

$$P_{el} = 0.84EC$$  \hspace{1cm} (19)

In Eq. (18), $C_z$ is the material factor, and $C_s$ is the structural imperfection factor. In Eq. (19), $C$ is the function of $\frac{t}{R_m}$ (as shown in Fig. 3), so $P_{el}$...
can be treated as the modified critical elastic buckling load \( P_e \).

2.5 RS2004

The calculation equation of RS2004\(^{[14]} \) is based on the work of Paliy\(^{[5]} \), it is also composed of failure pressure due to membrane yield (Eq.(20)) and failure pressure due to buckling (Eq. (21)).

\[
P = \frac{1}{s_t} P_y \tag{20}
\]

\[
P = \eta P_e \frac{s_t}{s_f} \tag{21}
\]

In Eq.(20) and Eq.(21), \( s_t \) is the safety factor, \( \eta \) is the modified coefficient due to nonlinear character of material and structural imperfection, as shown in Eq.(22) to Eq.(25).

\[
\eta = \eta_s \sqrt{1\left(1 + s_t f_s \right) \eta_0 \delta} \tag{22}
\]

\[
\delta = \frac{P_e}{\delta_y} \tag{23}
\]

\[
\eta_s = \frac{1}{1 + \left(2.8 + f_s \right)^2} \tag{24}
\]

\[
f_s = \frac{f}{t} \tag{25}
\]

In Eq.(25), \( f \) is the maximum deviation of spherical surface from regular round form.

2.6 GL2009

GL2009\(^{[11]} \) also calculates yielding load and buckling load. It divides the calculation pressure into three load cases: normal diving pressure, testing diving pressure and collapse diving pressure. All three load cases must be calculated and proofed respectively. It also uses modified yielding load \( P_y \) and modified critical elastic buckling load \( P_e \) in the calculation equations. They are similar to Eq.(7) and Eq.(8) and therefore these equations of GL2009 will not be repeated. It is interesting that GL2009 provides an advanced measurement method of the local flattening at spherical shells, as shown in Fig.4\(^{[11]} \).

2.7 ABS2010

ABS2010\(^{[7]} \) divides the calculation of the ultimate strength of spherical shell into two cases. When the shell is very thin, the critical elastic buckling load \( P_e \) of the shell is less than or equal to the yielding load \( P_y \), then
ABS2010 considers that the spherical shell will collapse due to buckling. As the thickness of the shell increases, when \( P_e \) is larger than \( P_y \), ABS2010 considers that the spherical shell will collapse due to the stress in the shell reaching the material’s ultimate strength, and the problem is translated to strength problems, as shown in Eq.(26) to Eq.(27).

\[
P = \frac{P_{cs}}{sf}
\]

\[
P_{cs} = \begin{cases} 
  P_y \cdot 0.7391 \left[1 + \left(\frac{P_y}{0.3P_e}\right)^2\right]^{1/2} & \text{for } P_e < 1 \\
  0.2124P_e & \text{for } P_e \geq 1
\end{cases}
\]

In Eq.(26), \( sf \) is the safety factor.

According the description of above 7 current available design rules, the classical yielding load \( (P_y) \) and the critical elastic buckling load \( (P_e) \) are the common foundation of the calculation of spherical shells. It can be easily found that \( P_y \) is equal to \( P_e \), when Eq.(28) is satisfied.

\[
t = \frac{\sigma_y \sqrt{3(1-\nu^2)}}{E}
\]

For Ti-6Li-4V titanium alloys, the curves of \( P_y \) and \( P_e \) are plotted in Fig.5. It can be found that if \( t/R_m \) is larger than 0.0117, \( P_e \) is larger than \( P_y \) and increases much faster than \( P_y \). This indicates that the problem will be translated from buckling to yielding, as the thickness of the shell increases.

![Fig.5 P_y versus P_e of titanium alloy](image)

### 3 Comparison of design rules

After the introduction of above seven available design rules, they are applied in the calcu-
lation of several existing spherical pressure hulls and the pressure hull of the new 4 500m manned submersible. In order to compare the calculation results, the input parameters of all design rules must be unified.

3.1 Unification of the input parameters

The input parameters of design rules can be divided into four kinds. The first is the dimension of the pressure hull which includes the radius of the spherical shell and the thickness of the shell. The second is the material properties which includes Young’s modulus, yield strength and Poisson’s ratio. The third is the structural imperfection which includes overall out-of-roundness (the maximum difference between the radius of actual shell and the design radius) and local shape deviation of the spherical surface of the shell from regular round form (call local out-of-roundness). The last is the additional coefficients which include uncertainty factor of material property, strength factor (safety factor), etc.

In this paper, the dimension of the pressure hull is chosen to be the design dimension of existing pressure hulls. The material properties are chosen to be the normal properties of titanium alloy (as given in Tab.1.), because the compared pressure hulls of existing submersible are all made of titanium alloy. Then, the structural imperfection (include overall-out-of-roundness and local out-of-roundness) is controlled not to exceed 0.5% × R, which meets the requirements of most current available design rules. At last, the safety factor is chosen to be consistent with the original design except the safety factor is provided by the rule directly.

3.2 Calculation results

In order to certificate the existing submersibles, the operating pressure of submersibles at diving depth must also be unified. Here the conversion formula of GL2009[11] is used which is given in Eq.(29).

\[ P \text{ (MPa)} = 0.0101 \times \text{Depth (m)} \] (29)

Based on the calculation equations of above seven rules and the chosen input parameters, the calculations were performed and the results are listed in Tab.2. From this table, it can be found that the calculation results of current available rules are significantly different. Moreover, the design thickness of most rules is much larger than the actual design thickness, except CCS1996[9] and RS2004[14].

Tab.2 Calculation results of existing pressure hulls

<table>
<thead>
<tr>
<th>Submersible</th>
<th>Actual design of existing pressure hull</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mir1 and Mir2</td>
<td>RUS (CONSUL)</td>
</tr>
<tr>
<td>Depth (m)</td>
<td>RUS (CONSUL)</td>
</tr>
<tr>
<td>6 000</td>
<td>6 000</td>
</tr>
<tr>
<td>Operating pressure from GL2009 (MPa)</td>
<td>60.6</td>
</tr>
<tr>
<td>Material</td>
<td>Martensite Ni steel</td>
</tr>
<tr>
<td></td>
<td>Titanium</td>
</tr>
<tr>
<td></td>
<td>Titanium</td>
</tr>
<tr>
<td></td>
<td>Titanium</td>
</tr>
<tr>
<td></td>
<td>Titanium</td>
</tr>
</tbody>
</table>
In Tab.3, a check of the fulfillment of pressure hulls of existing manned submersibles to these design rules is made, where \(\checkmark\) means pass and \(\times\) means cannot pass.  

**Tab.3  Check of actual designs of existing pressure hulls to the existing design rules**

<table>
<thead>
<tr>
<th>Submersible</th>
<th>Mir1 and Mir2 RUS (CONSUL)</th>
<th>Nautile</th>
<th>Shinhai 6500</th>
<th>Alvin (sf=1.55)</th>
<th>Alvin (sf=1.2)</th>
<th>Jiaolong</th>
</tr>
</thead>
<tbody>
<tr>
<td>DNV1988</td>
<td>(\checkmark)</td>
<td>(\checkmark)</td>
<td>(\checkmark)</td>
<td>(\checkmark)</td>
<td>(\checkmark)</td>
<td>(\times)</td>
</tr>
<tr>
<td>BV1989</td>
<td>(\times)</td>
<td>(\times)</td>
<td>(\times)</td>
<td>(\times)</td>
<td>(\times)</td>
<td>(\times)</td>
</tr>
<tr>
<td>LR1989</td>
<td>(\times)</td>
<td>(\times)</td>
<td>(\times)</td>
<td>(\times)</td>
<td>(\times)</td>
<td>(\times)</td>
</tr>
<tr>
<td>RS2004</td>
<td>(\checkmark)</td>
<td>(\checkmark)</td>
<td>(\checkmark)</td>
<td>(\checkmark)</td>
<td>(\checkmark)</td>
<td>(\checkmark)</td>
</tr>
<tr>
<td>CCS1996</td>
<td>(\checkmark)</td>
<td>(\checkmark)</td>
<td>(\checkmark)</td>
<td>(\checkmark)</td>
<td>(\checkmark)</td>
<td>(\checkmark)</td>
</tr>
<tr>
<td>ABS2009</td>
<td>(\times)</td>
<td>(\times)</td>
<td>(\times)</td>
<td>(\times)</td>
<td>(\times)</td>
<td>(\times)</td>
</tr>
<tr>
<td>GL2009</td>
<td></td>
<td>(\times)</td>
<td>(\times)</td>
<td>(\times)</td>
<td>(\times)</td>
<td>(\times)</td>
</tr>
</tbody>
</table>

*If the safety factor of 1.5 is used, then these two results are also not fulfilled.

These calculations show that many existing spherical pressure hulls are not in compliance with most current design rules. But the practical experiences of above existing manned submersibles have not been reported to meet any safety problems related to the load carrying capacity of spherical hulls.  

These rules were also applied to the design of the new 4 500m manned submersible, the same material properties in Tab.1 are used. The internal diameter of the spherical pressure hull is 2.0m which is the same as ALVIN and is designed to operate under 46.1MPa external pressure. With a safety factor of 1.5, the design results are given in Tab.4. All the results are higher than the actual thickness of ALVIN submersible and most of them are much thicker.  

**Tab.4 Design of new 4 500m manned submersible by different rules**

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>t (mm)</td>
<td>60.3</td>
<td>63.4</td>
<td>76.5</td>
<td>51.7</td>
<td>52.5</td>
<td>72.9</td>
<td>60.2</td>
</tr>
</tbody>
</table>

Both above significant differences among current design rules and the lack of agreement between actual hulls and the calculation of design rules pose a serious challenge to the validity of these design rules. First of all, only ABS2010 writes explicitly that its formulae can be
applied in the calculation of titanium spheres, other rules are limited to steel spheres and take titanium spheres as special case. Second, the number of manned submersibles which have been classified by current rules is too small, and more actual spherical pressure hulls should be provided to certification societies to validate and improve the formulae of current rules. Certainly this indicates that current design rules need to be updated and unified like Common Structural Rules for tankers and bulk carriers.[15]

Furthermore, from this comparison one may summarize the process of establishing a unified design rule. First, the safety problem of spherical pressure hull should be divided into buckling problem and strength problem, the corresponding formulas are Eq.(8) and Eq.(7). Then according to the analysis in section 2.8, the safety problem of most spherical pressure hull of manned submersible is strength problem, so Eq.(7) should be chosen as the basis for the calculation equation. As the basis equation is determined, additional formulae or factors which reflect the influences of structural imperfection, material uncertainty and safety factor need to be determined, and this work is in progress and will be reported in another paper. At last, the equations of new rules must be validated by applying them in the calculation of spherical pressure hulls of existing manned submersibles, and this work will be carried out after the determination of additional formulae and factors.

4 Summary and conclusions

A brief description of current available design rules of spherical pressure hull of submersible has been provided, and the common foundation of these rules was found to be either the classical yielding load formula or critical elastic buckling load formula. Then these rules were used to calculate the shell thickness of the spherical pressure hulls of several existing deep manned submersibles. In the calculations, the unification of input parameters is adopted because of the unavailability of the actual material properties for some submersibles. However, this will not be expected to affect the conclusions. From this comparison, significant differences among the calculation results of these rules have been found and the design thickness calculated by most rules is much larger than the actual design thickness of existing pressure hulls.

These calculations show that many existing spherical pressure hulls are not in compliance with most current design rules. But the practical experiences of above existing manned submersibles have not been reported to meet any safety problems related to the load carrying capacity of spherical hulls.

This poses a serious challenge to the validity of these design rules. Certainly this indicates that current design rules need to be updated and unified. The common basis of current design rules and the process of establishing a unified design rule have also been proposed.

References

深潜器载人球壳规范设计公式的比较

潘彬彬，崔维成
（中国船舶科学研究中心，江苏 无锡 214082）

摘要：文章首先简要地介绍了几个船级社现有的关于深潜器载人球壳的规范设计公式，这些公式背后的理论基础要么是经典的屈服破坏或屈曲破坏。然后，应用这些规范公式对国际上现有的几个载人潜水器耐压球壳的设计厚度进行了计算，并统一了输入的设计参数并将这些计算结果进行了比较。比较表明，不同船级社公式给出的球壳厚度存在很大差异，绝大多数设计规范计算出的厚度都超过潜水器的实际壳厚。因此，作者认为非常有必要对各船级社的规范设计公式进行更新，最好能和民船共同规范一样给出统一的球壳设计公式。最后，文中对如何建立统一公式的研究过程提出了建议。

关键词：耐压球壳；载人潜水器；设计规范；安全性
中国分类号：U662.1 文献标识码：A

作者简介：潘彬彬（1982-），男，中国船舶科学研究中心博士生，主要从事载人潜水器多学科优化设计方法研究；
崔维成（1963-），男，中国船舶科学研究中心研究员，博士生导师，主要从事载人潜水器的设计研究。
On Implementing a Practical Algorithm to Generate Fatigue Loading History or Spectrum from Short Time Measurement

LI Shan-shan, HUANG Zhi-bo, CUI Wei-cheng
(1. State Key Lab of Ocean Engineering, Shanghai jiao Tong University, Shanghai 200030, China; 2. China Ship Scientific Research Center, Wuxi 214082, China)

Abstract: Fatigue life prediction requires the information of fatigue loading history or spectrum over the whole design life. However, in many cases, the measured or simulated load history only includes a very short time compared to the entire design life. How to extrapolate the measured loads to longer time and how to count the information for every cycle are two important problems to be solved in fatigue analysis. In this paper, a method to extrapolate the measured loads to longer time is introduced together with the rainflow cycle counting method. Both methods have been implemented into computer programs. An example calculation is given to show the process. Problems on how to use these two methods to determine the design load are also pointed out.

Key words: load history; rainflow filter; load extrapolation; load spectrum; rainflow cycle; counting

CLC number: U662.2 Document code: A

1 Introduction

Fatigue is defined as a process of cycle by cycle accumulation of damage in a material undergoing fluctuating stresses and strains\textsuperscript{[1]}. A significant feature of fatigue is that the load is not large enough to cause immediate failure. Instead, failure occurs after a certain number of load fluctuations have been experienced, i.e. after the accumulated damage has reached a critical level. For marine structures such as ships and offshore platforms they are subjected to this type of cyclic loading, so fatigue is one of the most significant failure modes\textsuperscript{[2-4]}. Accurate prediction of the fatigue life of marine structures under service loading is very important for both safe and economic design and operation.

It is well-known that there are two types of fatigue life prediction methods\textsuperscript{[5]}. One is the cumulative fatigue damage (CFD) analysis where fatigue loading is expressed as a spectrum. The other is the fatigue crack propagation (FCP) analysis where fatigue loading is expressed as a time history. In this paper we distinguish the fatigue load history with the fatigue load spectrum while many other authors may not make this distinguish, e.g., Schijve\textsuperscript{[6]}. When we say a fatigue load
history, it means a time-domain distinction, e.g., \( P(t) \). When we say a fatigue spectrum, it means a frequency-domain distinction, e.g., Weibull distribution. The fatigue load history is used in fatigue crack propagation process while the fatigue load spectrum is used in cumulative fatigue damage calculation.

In fatigue life prediction of a structure, both the fatigue loading and fatigue strength are essential parameters. Hence, in order to obtain a proper fatigue design, it is crucial to consider real environmental loads. However, in many cases, the measured or simulated load history only covers a very short time, compared to the entire design life. Therefore, how to extrapolate the measured or calculated short term loads to longer time needs to be solved. The simplest method is to repeat one observed load block until the design life. This has the drawback that only the cycles in the measured signal will appear in the extrapolation, even though other cycles are also possible. In order to allow more extreme loads than the observed ones, statistical theories have to be employed. In Ref. [7], a statistical method is used that was proposed by Dressler et al. [8] where the rainflow matrix is extrapolated using kernel smoothing. An extrapolation method based on statistical extreme value theory [9] in combination with kernel smoothing is presented in Ref. [10].

In 2006, Johannesson [11] proposed an extrapolation method which can be applied to any signal and be used for any purpose. This method is implemented to study its performances and feasibility to be used for marine structures.

For fatigue life analysis, the load time history needs to be converted into cycle sequence or load spectrum. This can be done by choosing a suitable counting method. It is a common consensus that the rainflow cycle counting is the best [6, 12].

In this paper, a practical method for extrapolation of a measured time signal to a longer time period, based on statistical extreme value theory is first introduced. Then the method of rainflow cycle counting is introduced. Both methods have been implemented into computer programs. In order to demonstrate the application, the measurements of longitudinal strain in a deck position of a surface ship for 15 minutes [13] are used for extrapolation and rainflow cycle counting. Finally, issues to determine the design fatigue loading using the measured short history are pointed out for further study.

2 Generating longer loading history from short time measurement

2.1 Method for extrapolation of a load history

The methodology introduced here is updated from Ref. [11]. The main idea of this method is to repeat the measured load block, but modify the highest maxima and lowest minima in each block. The random regeneration of each block is based on statistical extreme value theory. An example showing the principle of the method is shown in Fig. 1, where three repetitions of a measured block are compared to three extrapolated blocks.
The horizontal dashed lines represent the threshold levels, $u_{min} = -6$ and $u_{max} = 6$, where the extrapolation starts (Ref.[11]).

The theoretical basis for the method is the so-called Peak Over Threshold (POT) technique in statistical extreme value theory, see Ref.[14]. Only the extreme excesses over a high level $u$ is modeled, i.e. the height of the excursions above $u$, see Fig.2. If we fix a high load level and study the excesses over this level, then under certain conditions these excesses approximately follow an exponential distribution for high enough threshold levels $u$. We then have the approximation for the excess $Z = \text{Max} \cdot u \in \text{Exp}(m)$, with cumulative distribution function

$$F(z) = 1 - \exp \left(- \frac{z}{m} \right), \quad m = \text{mean excesses over } u$$  \hspace{1cm} (1)

The estimation of the parameter in the exponential distribution is the sample mean of the excesses $z_1, \ldots, z_N$

$$m = \frac{1}{N} \sum_{i=1}^{N} z_i$$ \hspace{1cm} (2)

The family of possible distributions for excesses in extreme value theory is the Generalized Pareto Distribution (GPD),

$$F(z) = 1 - \left( 1 + \frac{\xi z}{\sigma} \right)^{-1/\xi},$$ \hspace{1cm} (3)

where the shape parameter $- \infty < \xi < \infty$ and the scale parameter $\sigma > 0$. When $\xi < 0$, the GPD has an upper endpoint, $0 < z < \sigma/\xi$, while for $\xi \geq 0$, $z > 0$. The special case of $\xi = 0$ is the exponential distribution. The estimation of the parameters in the GPD is known to be problematic, often giving large systematic errors, especially for small sample sizes. Care should be taken for the selection of moment method or maximum likelihood. Details on the maximum likelihood estimation can be found in Refs.[14, 27], methods based on moments are discussed in Ref.[28].

Mathematically the extreme value approximations are formulated as asymptotic results. For a random process $\{X(t): t \in [0, T]\}$ satisfying certain mixing conditions, it can be shown that the excesses over a level $u(T)$ converges in distribution to a GPD as $T \rightarrow \infty$ and $n(T)/T \rightarrow 0$, where $n(T)$ is the number of excesses over $u(T)$. The parameters of the limiting GPD de-
pend on the distributional properties of the process.

As a matter of fact, the exponential distribution of excesses is equivalent to modeling the global maximum as a Gumbel distribution, which works well in many applications. In this section, the model for excesses over a high level will be the exponential distribution, but it is straightforward to replace it by a GPD.

The generation of a k-fold extrapolated signal (sequence of turning points) can be performed in the following steps:

1. Start with a time signal.
2. Extract the turning points of the time signal, where the small cycles should be removed by using a rainflow filter.
3. Choose threshold levels $u_{\min}$ and $u_{\max}$ for the POT extrapolation. The choice of levels is discussed in section 2.2. Extract the excesses under $u_{\min}$ and the excesses over $u_{\max}$.
4. Estimate the mean excesses $m_{\min}$ and $m_{\max}$ under and over the thresholds $u_{\min}$ and $u_{\max}$, respectively.
5. Generate an extrapolated load block by simulating independent excesses as exponential random numbers. Replace each observed excess under $u_{\min}$ by a simulated exponential number with mean $m_{\min}$ and each observed excess over $u_{\max}$ by a simulated exponential number with mean $m_{\max}$.
6. Repeat step 5 until you have generated k extrapolated load blocks.
7. The k-fold extrapolated signal is obtained by putting the generated load blocks after each other.

From the k-fold extrapolated load time signal, we can also obtain the k-fold extrapolated load spectrum by applying the rainflow cycle counting algorithm which will be introduced in Section 3. In order to show the method for generating a k-fold extrapolated signal more clearly, a flow chart is given in Fig.3.

2.2 Choice of threshold levels

For the extrapolation, we need to choose the threshold levels where the extrapolation will start (see Fig.2). The choice of proper threshold levels is a difficult problem that demands some judgments. Note that the levels need to be chosen high enough for the extreme value theory to be a reasonable approximation, however also low enough to obtain a sufficient number of exceedances. Hence, it is important to verify that the exponential assumptions are fulfilled for the chosen threshold levels.

A useful diagnostic tool, for assessing the threshold, is the mean excess plot. The estimate of the mean exceedance over the threshold is plotted as a function of the threshold level (see Fig.4). For too low threshold levels, the extreme value approximation is not good enough, giving a systematic error, and for very high levels there is little information giving large scatter, which is seen through the confidence intervals. Hence, a proper level should be a compromise and be chosen somewhere in between in a region where the estimate is stable, i.e. the mean excess plot is approximately horizontal. Unfortunately, the mean excess plots may be hard to interpret, but they are still a valuable tool together with the exponential probability plots.
**Fig. 3** Flow chart of the generation of k-fold extrapolated signal

1. **Start with a time signal**
2. **Extract the turning points of the time signal**
3. **Remove the small cycles by using a rainflow filter**
4. **Choose threshold levels** $U_{\text{max}}$ and $U_{\text{min}}$ for POT extrapolation
   - (A default choice to find proper threshold levels such that there are about $\sqrt{N_0}$ exceedances, where $N_0$ is the number of cycles in the signal)
5. **Check if the choice is reasonable**
   - **Yes**: **Extract the excesses under** $U_{\text{min}}$ **and excesses over** $U_{\text{max}}$
   - **Estimate the mean excesses** $m_{\text{min}}$ and $m_{\text{max}}$, respectively
   - **Replace each observed excess under** $U_{\text{min}}$ **by a simulated exponential number with mean** $m_{\text{min}}$
   - **Replace each observed excess over** $U_{\text{max}}$ **by a simulated exponential number with mean** $m_{\text{max}}$
   - **Generate an extrapolated load block**
   - **k-fold extrapolation**
   - **Adjust the default choice of threshold levels according to checking result**

---

**PDF 文件使用 "pdfFactory Pro" 试用版本创建 www.fineprint.cn**
The choice of proper threshold levels is the most tricky part of the extrapolation. Here one needs to use experience together with the diagnostic tools to ensure that the exponential distribution is a good approximation. A default choice that seems to work well in many cases is to find threshold levels such that there are about $\sqrt{N_0}$ exceedances, where $N_0$ is the number of cycles in the signal.

3 Rain flow cycle counting

Cycle counting is the process of reducing a complex variable amplitude load history into a number of constant amplitude stress excursions, that can be related to available constant amplitude test data. This is a necessary step that needs to be carried out in order to predict fatigue crack growth in components subjected to variable amplitude loading. The method of cycle counting used often depends on the occurrences in the particular sequence that are considered to be significant in terms of fatigue damage [6, 12].

The definition of a cycle varies with the method of cycle counting. In fatigue analysis, a cycle is the load variation from the minimum to the maximum and then to the minimum load. At the moment, there is no frequency information contained in any cycle counting method but this can easily be added if three independent quantities are used. Furthermore, some of the counting methods can also give the information of the order of the cycles. Cycle counts can be made for all time histories such as force, stress, strain, torque, acceleration, deflection or other loading parameters of interest.

Cycle counting yields the marginal probability density distribution of the single quantity like amplitude or the joint probability density distribution of the pair such as (maximum, minimum) or (amplitude, mean) or (maximum, range).

Up to now, many different cycle counting methods have been developed such as level crossing counting, range counting and rainflow counting, see Refs. [6, 15]. One of the most important considerations in cycle counting is that the basis of the counting method needs to be com-
compatible with the understanding of the relevance of stress fluctuations to the fatigue process. Using this criterion, now it is almost the common consensus that the rainflow cycle counting method is the best cycle counting method.

The rainflow counting algorithm was first developed by Endo and Matsuiski in 1968. Downing and Socie created one of the more widely referenced and utilized rainflow cycle-counting algorithms in 1982, and which was included as one of many cycle-counting algorithms in ASTM E1049-85. Rychlik gave a mathematical definition for the rainflow counting method, thus enabling closed-form computations from the statistical properties of the load signal.

In principle, range counting includes counting of all successive load ranges, also small load variations occurring between adjacent larger ranges. It might be thought that small load variations can be disregarded in view of a negligible contribution to fatigue damage. A fundamental counting problem arises if a small load variation occurs between larger peak values. This situation is illustrated in Fig. 5. A two-parameter range counting procedure will count the ranges AB, BC and CD, and store this information in a matrix. Now, consider the situation that the intermediate range BC would not occur. Then, the large range AD would be counted only. Traditionally it was thought that fatigue damage is related to load ranges but now it is found that the maximum value is also important [e.g. Refs. 21, 22]. It should be expected that the fatigue damage of the large range AD alone is larger than for the three separate ranges AB, BC and CD. This has led to the so-called rainflow counting method of Endo and Matsuiski. The intermediate small load reversal BC is counted as a separate cycle and then removed from the major load range AD. This larger range can then be counted as a separate load range, see Fig. 5b. If four successive peak values are indicated by \( P_i, P_{i+1}, P_{i+2} \) and \( P_{i+3} \), the rainflow count requirement for counting and removing a small range from a larger range is:

\[
P_{i+1} < P_{i+3} \text{ and } P_{i+2} > P_i
\]  

(4)

If the intermediate small load reversal occurs in a descending load range, see Fig. 5c, the requirement is:

\[
P_{i+1} > P_{i+3} \text{ and } P_{i+2} < P_i
\]  

(5)

Fig. 5 Intermediate load reversal as part of a larger range.
In other words, the peak values of the intermediate small load reversal should be inside the range of the two peak values of the larger range. Successive rainflow counts are indicated in Fig. 6. In Fig. 6a five rainflow counts can be made. After counting and removing these small cycles, Fig. 6b is obtained. In this figure again three rainflow counts can be made, but now of larger ranges. Removing these cycles leads to Fig. 6c in which again two still larger load reversals can be counted and removed. In the final residue, Fig. 6d, no further counts are possible. The ranges of the residue must be counted separately at the end of the counting procedure. The rainflow count results can be stored in a two-parameter matrix.

The rainflow count procedure has found some support by considering cyclic plasticity. A short load sequence is given in Fig. 7a, which leads to counting two intermediate load reversals by the rainflow count method, as indicated in this figure. The corresponding plastic behavior is schematically indicated in Fig. 7b, which could be applied to local plasticity at the material surface during the initiation period, or to crack tip plasticity during crack growth. The intermediate load reversals C1 and C2 are causing hysteresis loops inside the major hysteresis of the major cycle between A and B. It is thus assumed that the intermediate plasticity loops do not affect the major loop. This reasoning gives somewhat speculative support to the rainflow counting method.

The rainflow counting method can also be illustrated in Fig. 8. The original method was so-called because, if the stress or strain history is presented vertically, the algorithm can be considered analogous to the behavior of rain flowing from a pagoda style roof.

The rules for this method are as follows:

1. Read the next peak or valley. If out of data go to Step (6).
2. If there are fewer than three points go to Step (1). Form ranges X and Y using the three
most recent peaks and troughs that have not been discarded.

(3) Compare the absolute values of ranges $X$ and $Y$.

(a) If $X < Y$, go to Step (1).
(b) If $X \geq Y$, go to Step (4).

(4) If range $Y$ contains the starting point $S$, go to Step (5); otherwise, count range $Y$ as one cycle; discard the peak and trough of $Y$; and go to Step (2).

(5) Count range $Y$ as one half cycle; discard the first point (peak or trough) in range $Y$; move the starting point to the second point in range $Y$; and go to Step (2).

(6) Count each range that has not previously been counted as one half cycle.

These rules may be converted to an algorithm suitable for computation. Commercial program for rainflow cycle counting is also available now, e.g. Durability Rainflow Cycle Counting 2002\textsuperscript{[23, 24]} In this paper, a named “RAINFLOW” code developed by Nieslony\textsuperscript{[24]} is used. It first searches turning points from signals and then carries out rainflow cycle counting according to the algorithm mentioned above. The range and maximum of every cycle can be obtained by an improved edition of this code.

4 A practical example of extrapolation and rainflow cycle counting of a load history

In order to demonstrate the extrapolation and rainflow cycle counting methods introduced above, a full scale test of wave bending moments on a surface ship is used. Fig.9 shows the measurements of the longitudinal strain in a deck position of a surface ship for 15 minutes\textsuperscript{[13]}. 

![Fig.8 The rainflow cycle counting method\textsuperscript{(1)]]}

![Fig.9 The measured load signals\textsuperscript{(13)]]}
For this example, we first choose the first three minutes to be extrapolated to a 5 times as a longer time period and compare with the original measurement. First the turning points of the time signal are extracted. Further the signal should be rainflow filtered in order to remove small oscillations that do not contribute to the fatigue damage. It is also advantageous for the POT analysis, since it removes small oscillations during excursions. A default choice for the rainflow filter is to set the threshold range to 5% of the total range of the signal, see Fig. 10. Then the threshold levels \( u_{\text{min}} \) and \( u_{\text{max}} \) for the POT extrapolation are chosen. We choose 121 \( \mu \text{e} \) and -111 \( \mu \text{e} \) as \( u_{\text{max}} \) and \( u_{\text{min}} \), respectively, see Fig.11. Here we use a default choice to find threshold levels such that there are about \( \sqrt{N_0} \) exceedances, where \( N_0 \) is the number of cycles in the signal. For assessing if the thresholds are proper, we use the mean excess plot method to check it, the result is shown in Fig.12, where the chosen threshold levels are in the regions where the estimate is almost stable. With the measured time period increasing, e.g. 15 minutes chosen to be extrapolated in this example, the horizontal region in Fig.12 is more clear. The comparison between measured time signal and extrapolated time signal for 15 minutes is shown in Fig.13.
Fig. 12 Mean excess plot

Check result of upper threshold levels

Check result of lower threshold levels

Fig. 13 A comparison between measured time signal and extrapolated time signal for 15 minutes

Let us apply the rainflow cycle counting method to the two load-time series shown in Fig. 13. For each time series, we will obtain a certain number of complete cycles and some half cycles. For these half cycles, every two half cycles are merged as an equivalent complete cycle and they are put in the end of the series of cycles. For each cycle, we use both maximum and minimum values to represent it. The counted results are shown in Fig. 14.

For the counted maximum values and the ranges, they can be plotted as a spectrum. Fig. 15a shows a comparison of the load spectra of the 5-fold extrapolations to the 15 minutes of the measured ones for the maximum value of each cycle while Fig. 15b shows the range. As is customary, the load spectra are plotted as the cumulative number of cycles larger than a given amplitude as function of the amplitude. We observe that the extrapolation looks reasonable since it agrees well with the observed load spectrum. Further, the load spectra are extrapolated
to higher amplitudes, and are smoother compared to the observed ones. These are the properties of the recommended extrapolation method.

![Graph showing original measurement and extrapolated signal]

Fig. 14 Rainflow cycle counted results for two load time signals given in Fig. 13 (threshold range to 5% of the total range)

(a) Original measurement 274 cycles
(b) Extrapolated time signal 294 cycles

![Graph showing maximum values and ranges of cycles]

Fig. 15 Five-fold extrapolated load spectra compared to the measured load spectra

Now let us take the original measurement of 15 minutes as the source data for extrapolation and to check the effect of folds on the distribution properties. The extrapolation method is based on a random simulation of the high maxima and low minima. Hence, the resulting extrapolated turning points, and hence also the load spectra, will be different from each new simulation. As the number of simulations or the number of folds increases, the load spectra will converge. Fig. 16 shows the load spectra for the range of cycles for five different fold numbers: 1-fold (original measurement of 15 minutes), 4-fold (1 hour), 96-fold (1 day). Due to the limit of hard disk, longer extrapolations are not carried out here.

![Graph showing load spectra for different fold numbers]

Fig. 16 Effect of fold number on the load spectra for the range of cycles
5 Further issues for the determination of design load from the short time measurement

How to construct the design load from limited practical measurements is a very important issue. However, from Fig.16 one may be aware that using different measurement results may result in different design loads if we simply apply the method introduced above. This is certainly not satisfactory from practical application point of view. In order to overcome this deficiency, we suggest to take into account the corresponding wave data information. In order to determine the design load for a specific ship, we first need to provide the time fraction for each sea state, \((S_i, p_i)\), where \(i\) is the sea state number from 0 to 12, 0 can be used as the port time, \(S_i\) is the \(i\)th sea state number and \(p_i\) is the time fraction of the \(i\)th sea state. Now if we know a short time period of sea state \(k\). We can convert this measurement to different sea states assuming all the responses are in linear range. Then for each sea state, we extrapolate to the corresponding fraction of the design life and count the cycles. Summarizing all the cycles for different sea states, we can derive the load spectra. This load spectrum will be independent of the short time measurements. If the time fraction for each sea state, \((S_i, p_i)\) has been specified, no matter what sea state is used for the short time measurement, the same design load spectrum will be derived. However, how to construct the time sequence is another challenging problem. This is basically the problem to construct a so-called standardized load history. This problem will be investigated in the future.

6 Summary and conclusions

Fatigue life prediction requires the information of fatigue loading history or spectrum over the whole design life. There are basically two types of methods to determine the fatigue loads, one is based on calculation and the other is based on experiment. However, in many cases, the measured load history can only include a very short time compared to the entire design life. How to extrapolate the measured loads to longer time and how to count the information for every cycle are two important problems to be solved in fatigue analysis. In this paper, a method to extrapolate the measured loads to longer time has been introduced together with the rainflow cycle counting method. Both methods have been implemented into computer programs. An example calculation is given to illustrate the process. Problems on how to use these two methods to determine the design load are also pointed out.

Acknowledgements

This study was supported by the Innovative Scholars Support Program of Jiangsu Province, Project No. SBK 2008004, 2008-2010.
References


从短期测量数据来生成疲劳载荷的一种实用算法的开发

李珊珊1, 黄志博2, 崔维成1,2

(1 上海交通大学海洋工程国家重点实验室, 上海 200030; 2 中国船舶科学研究中心, 江苏 无锡 214082)

摘要: 疲劳寿命预报需要知道在全设计寿命期内的疲劳载荷时间历程或载荷谱的信息。但是, 在很多情况下, 实际测量或模拟的载荷时间历程与设计寿命相比只包含很短的一段时间。如何将测量的数据外插到更长的时间段以及如何计数出每周的信息是疲劳分析中必须要解决的两个重要问题。本文的主要目的就是介绍一种将实测载荷外插的方法以及雨流计数法。这两种方法均被编成相应的计算程序。一条水面船的实测数据用作例子来演示计算过程。最后对如何采用这两种方法来构建疲劳设计载荷问题也提出了思路。

关键词: 载荷历程; 雨流过滤器; 载荷外插; 载荷谱; 雨流计数

中图分类号: U662.2 文献标识码: A

Effect of Hydrostatic Pressure on Input Power Flow in Submerged Ring-stiffened Cylindrical Shells

LIU Zhi-zhong, LI Tian-yun, ZHU Xiang, ZHANG Jun-jie

(Department of Naval Architecture and Ocean Engineering, Huazhong University of Science and Technology, Wuhan 430074, China)

Abstract: The input power flow for an infinite ring-stiffened cylindrical shell submerged in fluid induced by a cosine harmonic circumferential line force under conditions of a uniform external hydrostatic pressure field is investigated in this paper. The motion of the shell and the pressure field of the external fluid are described by the Flügge’s thin shell theory and the Helmholtz equation respectively. The effect of the external pressure field is modeled by including static prestress terms in the shell equations of motion. The effects of hydrostatic pressure on the input power flow are examined. The results show that the external pressure shifts the curves of input power flow to left along the frequency axis. The effect is more obvious with higher external pressure and circumferential mode. It will give some guidelines for vibration and noise control of this kind of shell.

Key words: hydrostatic pressure; input power flow; ring-stiffened shell; submerged shell

1 Introduction

A submerged cylindrical shell reinforced by circumferential rings is the primary structure of submarine, torpedo and all kinds of submerged vehicles. The determination of the dynamic response of the structures is thus a significant subject in vibration and noise control area. The method of vibration power flow is an effective tool used in this area, and the concept of power flow was first definitely presented by Goyder and White[1-3]. Zhang and Zhang[4] have introduced the concept of power flow into the analysis of periodic shells, and studied the input vibrational power flow from a cosine harmonic circumferential line force and the power transmitted by internal forces of the shell wall in vacuo.

Fuller[5] has investigated the input mobility of an infinite elastic circular cylindrical shell filled with fluid. The spectral equations of motion of the shell-fluid system and the method of residues were employed to evaluate the mobility, and their physical interpretation was also discussed. Xu et al.[6-7] have studied the vibration power flow input and transmission in a fluid-filled cylindrical shell. A technology of the spatial Fourier transforms and the inverse transforms was applied in their studies.
Practically, a cylindrical shell submerged in water is always reinforced by rings or/and bulkheads. Mead and Bardell\cite{8} researched free vibration of a thin cylindrical shell with periodic circumferential stiffeners. When periodic structure is surrounded by the fluid field, the space harmonic analysis method presented by Mead\cite{9} has been widely used. Using this method, Xu\cite{10} investigated the input power flow in a periodically stiffened shell filled with fluid. Yan\cite{11-13} also studied the characteristics of the vibrational power flow propagation and sound radiation in an infinite submerged periodic ring-stiffened cylindrical shell by the same method. Burroughs\cite{14} studied the fluid-loaded infinite circular cylindrical with doubly periodic ring supports forced by a point excitation and gave the analytical expression of far field acoustic radiation, which established the basal thought of this kind of problem. But the model he used is comparatively simple, only the normal force between the stiffeners and the shell was considered.

When the structure is located in a dense medium such as water, there is a resultant stress field in the structure even though exists in the absence of any vibrations or acoustic loadings. The presence of this static additional stress state in the structure changes the structural response characteristics including, for example, the natural frequencies of vibration\cite{15-16} and acoustic response of the structure\cite{17}. These effects will then subsequently result in a variation of forced vibration response of the structure as well.

The purpose of the present study is to include the hydrostatic pressure field effects in the formulation of the coupled vibro-acoustic response of submerged ring-stiffened cylindrical shells. There exists an extensive literature discussing the dynamic response of prestressed shells\cite{18-20}. Xie and Luo\cite{21} studied the acoustic radiation properties of ring-stiffened cylindrical shells submerged in fluid by means of Hamilton’s principle and Green function, and the effects of hydrostatic pressure and rings on the acoustic radiation of the shells were also discussed.

In this paper, the space harmonic analysis is extended to investigate the input power flow induced by a cosine harmonic line force for an infinite ring-stiffened cylindrical shell submerged in water. The effects of hydrostatic pressure on the input power flow are examined.

2 Forced vibration of the coupled system

An infinite thin-walled periodic ring-stiffened cylindrical shell submerged in fluid is considered, as shown in Fig.1. The shell is characterized by its mean radius $R$, wall thickness $h$, mass density $\rho_s$, Young’s modulus $E$ and the Poisson’s ratio $\mu$. The density of the external fluid is $\rho_f$ and the sound velocity in it is $C_f$. The ring stiffeners have uniform rectangular section with width $b$ and height $d$, attached at $x= mL$ ($L$ is the stiffener spacing, $m=0, \pm1, \pm2, \cdots$). The connections are rigid, so that, at each line of attachment, the shell and the stiffeners have the same linear velocity and angular velocity. The inside stiffeners may apply axial force, shear and moments on the shell. To simplify the problem, it is assumed that the stiffeners are located on the inner wall of the shell, so that their interaction with the external fluid can be ignored.
2.1 The motion equations of the periodic shell

The shell is excited by a harmonic line pressure $F$, acting on $x=0$, expressed as

$$F(\theta, t) = F_0 \cos(n\theta) \delta(t) \exp(i\omega t)$$

where $F_0$ is amplitude of the pressure, $\omega$ is the circular frequency, $n$ is the circumferential model order and $\delta$ is the Dirac delta function.

The cylindrical coordinate system $(x, \theta, r)$ is adopted in the analysis shown in Fig.1. The equations of structure motion are taken from Flugge's thin shell theory\(^2\) as

$$u_{xx} + \frac{1}{2} \mu u_{\theta \theta} + \frac{1}{2} \mu v_{xx} + \mu w_x + \frac{1}{2} \mu w_{xx} + \frac{1}{2} \mu w_{\theta \theta} + \frac{1}{2} \mu w_{\theta \theta}$$

$$u_{x}x + \frac{1}{2} \mu u_{\theta \theta} + \frac{1}{2} \mu v_{xx} + \mu w_x + \frac{1}{2} \mu w_{xx} + \frac{1}{2} \mu w_{\theta \theta} + \frac{1}{2} \mu w_{\theta \theta}$$

$$T_1 u_{xx} + T_2 (u_{\theta \theta} - w_x)$$

$$= F_u,$$  \hspace{1cm} (2a)

$$+ T_1 u_{\theta \theta} + \frac{1}{2} \mu u_{xx} + \frac{1}{2} \mu v_{xx} + w_x + K \left( \frac{3}{2} \mu v_{xx} + \frac{3}{2} \mu w_{xx} \right) + T_1 v_{xx} + T_2 (v_{\theta \theta} + w_\theta)$$

$$= F_v,$$  \hspace{1cm} (2b)

$$+ \frac{1}{2} \mu w_{xx} + T_1 w_{xx} + T_2 (u_{xx} - v_{\theta \theta} + \frac{1}{2} \mu w_{xx} + \frac{1}{2} \mu w_{\theta \theta})$$

$$+ 2K w_{\theta \theta} - T_1 w_{xx} - T_2 (u_{xx} - v_{\theta \theta} + w_{\theta \theta})$$

$$+ \frac{1}{2} \mu w_{xx} + T_1 w_{xx} + T_2 (u_{xx} - v_{\theta \theta} + w_{\theta \theta})$$

$$= F_w,$$  \hspace{1cm} (2c)

where, $(\cdot)_x = R \frac{\partial}{\partial x}$, $(\cdot)_\theta = \frac{\partial}{\partial \theta}$, $(\cdot)_t = \frac{\partial}{\partial t}$; $K = h^2 / 12R^2$; $T_1$ and $T_2$ are the terms containing the effects of additional structural stress from the hydrostatic pressure field, $T_1 = (R/2 Eh) \left( 1 - \mu^2 \right) p_0$, $T_2 = (R/E h) \left( 1 - \mu^2 \right) p_0$, $p_0$ is external hydrostatic pressure, of which the effects are modeled by including static prestress terms reflecting the stress state that would be present in a cylindrical pressure vessel. These stresses consist of an additional axial stress component and an additional hoop stress component, and are assumed to exist prior to an independent of the dynamic stresses arising from vibration; $F_u$, $F_v$ and $F_w$ are the external loads exerted on the shell wall in the $x$, $\theta$ and $r$ directions respectively, which are given by

$$F_u = \sum_{n=-\infty}^{\infty} F_{u, n} \delta(x - mL)$$  \hspace{1cm} (3a)
\[ F_v = D \sum_{m=-\infty}^{\infty} F_{v,m} \delta(x-mL) \]  \hspace{1cm} (3b)

\[ F_w = D \left[ (F - P_f) + \sum_{m=-\infty}^{\infty} F_{w,m} \delta(x-mL) + \sum_{m=-\infty}^{\infty} M_m \delta'(x-mL) \right] \]  \hspace{1cm} (3c)

where, \( D = R^2 \left| 1 - \mu^2 \right| / \varepsilon \), \( \delta' = \delta / \delta x \), \( P_f \) is the acoustic pressure, \( F_{u,m}, F_{v,m}, F_{w,m} \) and \( M_m \) are the sideeward forces and moments of the \( m \)th stiffener acting on the shell, respectively.

2.2 Fluid acoustic equations

The fluid is assumed to be non-viscous, isotropic and irrotational which satisfies the acoustic wave equation. The equation of motion of the fluid can be written by Helmholtz equation in the cylindrical coordinate system \((x, \theta, r)\) as

\[ \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial P_f}{\partial r} \right) + \frac{1}{r^2} \frac{\partial^2 P_f}{\partial \theta^2} + \frac{\partial^2 P_f}{\partial x^2} = \frac{1}{C_f^2} \frac{\partial^2 P_f}{\partial t^2} \]  \hspace{1cm} (4)

where the \( x, \theta \) and \( r \) coordinate are the same as those of the shell.

Applying variables separation method to solve the acoustic wave equation, the associated form of the pressure field is expressed as

\[ P_f = \sum_{n=0}^{\infty} \sum_{s=1}^{\infty} P_{ns} \cos(n\theta) H_n^2(k_{fr}^r) \exp[i\omega t - ik_x x] \]  \hspace{1cm} (5)

where subscript \( s \) denotes a particular branch of the dispersion curve; \( P_{ns} \) is the fluid acoustic pressure amplitude of every \( n \) and \( s \); \( H_n^2(k_{fr}^r) \) is Hankel function of the second kind and order \( n \); \( k_r^r \) and \( k_x \) are the radial and axial wave numbers respectively, and their relation is

\[ k_r^2 = k_x^2 \]  \hspace{1cm} (6)

where \( k_r \) is the free wave number, \( k_r = \omega / C_r \).

The displacement components of the shell are expressed in a travelling wave form as

\[ u = \sum_{n=0}^{\infty} \sum_{s=1}^{\infty} U_{ns} \cos(n\theta) \exp(i\omega t - ik_x x) \]  \hspace{1cm} (7a)

\[ v = \sum_{n=0}^{\infty} \sum_{s=1}^{\infty} V_{ns} \sin(n\theta) \exp(i\omega t - ik_x x) \]  \hspace{1cm} (7b)

\[ w = \sum_{n=0}^{\infty} \sum_{s=1}^{\infty} W_{ns} \cos(n\theta) \exp(i\omega t - ik_x x) \]  \hspace{1cm} (7c)

where \( U_{ns}, V_{ns} \) and \( W_{ns} \) are the displacement amplitudes in the \( x, \theta \) and \( r \) directions respectively.

As usual, the fluid velocity is continuous across the fluid-shell boundary, leading to the boundary condition

\[ \left. \frac{1}{i\omega} \frac{\partial P_f}{\partial t} \right|_{r=R} = \left. \frac{\partial N}{\partial t} \right|_{r=R} \]  \hspace{1cm} (8)
Substituting Eqs.(5) and (7c) into Eq.(8), the fluid acoustic pressure amplitude $P_{ns}$ is obtained as

$$P_{ns} = \left[ \frac{2 \omega \rho_f / (k_n^2 H_n^2' / (k_n^2 R))}{k_n^2 H_n^2' / (k_n^2 R)} \right] W_{ns}$$

(9)

Introducing Eq.(9) into Eq.(5), acoustic pressure can be obtained as

$$P_f = \sum_{n=0}^{\infty} \sum_{s=1}^{\infty} \frac{2 \omega \rho_f H_n^2 / (k_n^2 R)}{k_n^2 H_n^2' / (k_n^2 R)} W_{ns} \cos(n \theta) \exp(i \omega t - k_n^2 x)$$

(10)

2.3 The response of the coupled system

Introducing Eqs.(7), (10) into Eq.(2) and taking the Fourier transform, the following equations are obtained in the matrix form as

$$[L_{3x3}] \begin{bmatrix} \tilde{U} \\ \tilde{V} \\ \tilde{W} \end{bmatrix} = \begin{bmatrix} F_u & F_v & F_w \end{bmatrix} \begin{bmatrix} \tilde{F}_U \\ \tilde{F}_V \\ \tilde{F}_W \end{bmatrix}$$

(11)

where $\tilde{U}$, $\tilde{V}$ and $\tilde{W}$ are the spectral displacements,

$$\tilde{F} = D \int_{-\infty}^{\infty} F(x) e^{ikx} \delta(k-k_v) dx = DF_0 \delta(k-k_v)$$

(12a)

$$\tilde{F}_u = D \int_{-\infty}^{\infty} F_{u,m} \delta(x-mL) e^{ikx} dx = D \sum_{m=0}^{\infty} F_{u,m} e^{ikmL}$$

(12b)

$$\tilde{F}_v = D \int_{-\infty}^{\infty} F_{v,m} \delta(x-mL) e^{ikx} dx = D \sum_{m=0}^{\infty} F_{v,m} e^{ikmL}$$

(12c)

$$\tilde{F}_w = D \int_{-\infty}^{\infty} F_{w,m} \delta(x-mL) e^{ikx} dx = D \sum_{m=0}^{\infty} F_{w,m} e^{ikmL}$$

(12d)

$$\tilde{M} = D \int_{-\infty}^{\infty} M_{m} \delta'(x-mL) e^{ikx} dx = DK e^{ikmL}$$

(12e)

where $k$ is axial wave number in spectral domain.

In Eq.(11), $|L_{3x3}| = \begin{bmatrix} L_{11} & L_{12} & L_{13} \\ L_{12} & L_{22} & L_{23} \\ L_{13} & L_{23} & L_{33} \end{bmatrix}$ and the elements are given by

$$L_{11} = i \omega \sqrt{\rho_f R^2 (1+\mu^2) / E}$$

$$L_{12} = i \lambda \mu / 2$$

$$L_{13} = i \left( (\mu - T_2) \lambda + K \lambda^2 - K (1+\mu) \lambda n^2 / 2 \right)$$

$$L_{22} = (1+3K) \lambda^2 / 2$$

$$L_{23} = (1+T_2) n K \lambda^2 / 2$$

$$L_{33} = (1+K) \lambda^4 + 2K n \lambda^2 + K n^2 (2K - T_2) n^2 + T_2 \lambda^2 - \Omega^2$$

where $\Omega$ is non-dimensional frequency, $\Omega = \omega \sqrt{\rho_f R^2 (1+\mu^2) / E}$; $\lambda$ is non-dimensional axial
wavenumber, \( \lambda = k_x R \); FL is fluid-loading term, and is given by

\[
\mathcal{F}_L = \frac{\rho_l R^2}{\rho_s H} \frac{H_0^2(k R)}{H_0^2(k R)}
\]

(13)

As the rings are periodic in x-axes direction, the forces in the stiffeners acting on the shell wall satisfy the periodicity condition as

\[
[F_{u,0} \ F_{v,0} \ F_{w,0} \ M_0]^T = e^{-i k m L} [F_{u,0} \ F_{v,0} \ F_{w,0} \ M_0]^T
\]

(14)

where \( F_{u,0}, F_{v,0}, F_{w,0} \) and \( M_0 \) are the forces or moments of 0th \((m=0)\) stiffener.

By using Poisson sum formula\(^{[23]}\), the following equations can be obtained:

\[
\begin{align*}
\tilde{F}_u &= 2 \pi D F_{u,0} \sum_{m=-\infty}^{\infty} \delta(2m \pi + k_x R) L \\
\tilde{F}_v &= 2 \pi D F_{v,0} \sum_{m=-\infty}^{\infty} \delta(2m \pi + k_x R) L \\
\tilde{F}_w &= 2 \pi D F_{w,0} \sum_{m=-\infty}^{\infty} \delta(2m \pi + k_x R) L \\
\tilde{M} &= i \pi D k_x M_0 \sum_{m=-\infty}^{\infty} \delta(2m \pi + k_x R) L 
\end{align*}
\]

(15a)

(15b)

(15c)

(15d)

Let matrix \( I \) be the inverse of matrix \( L \), then the spectral displacements can be obtained from Eq.(11) as follows

\[
\begin{bmatrix}
\tilde{U} \\
\tilde{V} \\
\tilde{W}
\end{bmatrix} =
\begin{bmatrix}
I_{11} & I_{12} & I_{13} \\
I_{21} & I_{22} & I_{23} \\
I_{31} & I_{32} & I_{33}
\end{bmatrix}
\begin{bmatrix}
\tilde{F}_u \\
\tilde{F}_v \\
\tilde{M} + \tilde{F}_w
\end{bmatrix}
\]

(16)

Taking the inverse Fourier transform to Eq.(16), the shell displacement can be obtained as:

\[
\begin{align*}
\frac{U(x)}{D} &= \frac{F_{u,0}}{2 \pi} I_{13} (k_x) \exp(-i k_m x) + F_{u,0} \sum_{m=-\infty}^{\infty} \left[ \exp(-i k_m x) I_{12} (k_m) \right] \\
&\quad + F_{v,0} \sum_{m=-\infty}^{\infty} \left[ \exp(-i k_m x) I_{13} (k_m) \right] \\
&\quad - i M_0 \sum_{m=-\infty}^{\infty} \left[ \exp(-i k_m x) k_m I_{13} (k_m) \right] \\
\frac{V(x)}{D} &= \frac{F_{v,0}}{2 \pi} I_{23} (k_x) \exp(-i k_m x) + F_{u,0} \sum_{m=-\infty}^{\infty} \left[ \exp(-i k_m x) I_{12} (k_m) \right] \\
&\quad + F_{v,0} \sum_{m=-\infty}^{\infty} \left[ \exp(-i k_m x) I_{22} (k_m) \right] + F_{w,0} \sum_{m=-\infty}^{\infty} \left[ \exp(-i k_m x) I_{23} (k_m) \right] \\
&\quad - i M_0 \sum_{m=-\infty}^{\infty} \left[ \exp(-i k_m x) k_m I_{23} (k_m) \right]
\end{align*}
\]
where, $k_m\equiv k_x + 2m\pi/L$.  

Thus, the space displacement at $x=0$ is given by:

\[ u(0) = \frac{DF_0}{2\pi} \sum_{m=0}^{\infty} k_m I_{13}(k_m) F_{u,0} + \sum_{m=0}^{\infty} I_{11}(k_m) F_{v,0} \sum_{m=0}^{\infty} I_{12}(k_m) F_{w,0} \sum_{m=0}^{\infty} I_{13}(k_m) \]

\[ -iDM_0 \sum_{m=0}^{\infty} k_m I_{13}(k_m) \]  

(17a)

\[ v(0) = \frac{DF_0}{2\pi} \sum_{m=23}^{\infty} k_m I_{23}(k_m) F_{u,0} + \sum_{m=0}^{\infty} I_{22}(k_m) F_{v,0} \sum_{m=0}^{\infty} I_{23}(k_m) F_{w,0} \sum_{m=0}^{\infty} I_{23}(k_m) \]

\[ -iDM_0 \sum_{m=0}^{\infty} k_m I_{23}(k_m) \]  

(17b)

\[ w(0) = \frac{DF_0}{2\pi} \sum_{m=33}^{\infty} k_m I_{33}(k_m) F_{u,0} + \sum_{m=0}^{\infty} I_{32}(k_m) F_{v,0} \sum_{m=0}^{\infty} I_{33}(k_m) F_{w,0} \sum_{m=0}^{\infty} I_{33}(k_m) \]

\[ -iDM_0 \sum_{m=0}^{\infty} k_m I_{33}(k_m) \]  

(17c)

Once the forces and moments of the 0th ($x=0$) stiffener in Eq.(17) are given, the shell response to the applied pressure load $F$ can be easily solved. Applying the compatible condition of the forces and the displacements between the shell and the stiffeners at $x=0$, the reaction forces can be expressed as follows\cite{10}:

\[ F_{w,0} = K_1 w(0) - K_3 v(0), \quad F_{v,0} = K_2 w(0) + K_3 v(0) \]  

(18a, b)

\[ F_{u,0} = K_4 u(0) + (K_5 - \frac{e_i}{R} K_4) (-iK_1 R w(0))) \]  

(18c)

\[ M_0 = K_4 u(0) + (K_6 - \frac{e_i}{R} K_5) (-iK_1 R w(0)) \]  

(18d)

where, $e_i$ is the distance from the midsurface of the shell to the geometric center of the stiffener, $K_1$-$K_6$ are given in the Ref.[10].

Introducing Eq.(18) into Eq.(17), the shell displacement at $x=0$ can be obtained in matrix form as

\[ H_{3\times3} \begin{bmatrix} u(0) \\ v(0) \\ w(0) \end{bmatrix} = \begin{bmatrix} F^u \\ F^v \\ F^w \end{bmatrix} \]  

(19)
where \( F^u = \frac{F_0}{2\pi} l_{13}, \) \( F^v = \frac{F_0}{2\pi} l_{23}, \) \( F^w = \frac{F_0}{2\pi} l_{33}, \) and the elements of matrix \( H \) are given by:

\[
H_{11} = \frac{1}{D} - K_{4} J_{11} + i K_{5} K_{13}, \quad H_{12} = K_{3} J_{12} + K_{2} J_{13}, \quad H_{13} = i k_x \left( k_5 - e_k K_0 / R \right) J_{11} - K_{2} J_{22} + K_{3} J_{13} + k_x \left( K_6 - e_k K_0 / R \right) K_{13}, \quad H_{21} = K_{4} J_{12} + i K_{5} K_{23}, \quad H_{22} = \frac{1}{D} - K_{3} J_{22} + K_{2} J_{23}, \quad H_{23} = i k_x \left( K_5 - e_k K_0 / R \right) J_{12} - K_{2} J_{22} + K_{3} J_{23} + k_x \left( K_6 - e_k K_0 / R \right) K_{23}, \quad H_{31} = K_{4} J_{13} + i K_{5} K_{33}, \quad H_{32} = K_{3} J_{23} + K_{2} J_{33}, \quad H_{33} = \frac{1}{D} - i k_x \left( K_5 - e_k K_0 / R \right) J_{13} - K_{2} J_{23} + K_{3} J_{33} + k_x \left( K_6 - e_k K_0 / R \right) K_{33}.
\]

where \( J_{11} = \sum_{m=1}^{\infty} l_{11} \left( k_m \right), \) \( J_{12} = \sum_{m=1}^{\infty} l_{12} \left( k_m \right), \) \( J_{13} = \sum_{m=1}^{\infty} l_{13} \left( k_m \right), \) \( J_{22} = \sum_{m=1}^{\infty} l_{22} \left( k_m \right), \) \( J_{23} = \sum_{m=1}^{\infty} l_{23} \left( k_m \right), \) \( J_{33} = \sum_{m=1}^{\infty} l_{33} \left( k_m \right), \)

\[
K_{13} = \sum_{m=1}^{\infty} k_m l_{13} \left( k_m \right), \quad K_{23} = \sum_{m=1}^{\infty} k_m l_{23} \left( k_m \right), \quad K_{33} = \sum_{m=1}^{\infty} k_m l_{33} \left( k_m \right).
\]

From Eq.(19), the displacements of the stiffened shell at \( x=0 \) can be easily solved as follows

\[
\begin{bmatrix} u(0) \\ v(0) \\ w(0) \end{bmatrix}^T = Q_{3 \times 3} \begin{bmatrix} F^u & F^v & F^w \end{bmatrix}^T \tag{20}
\]

where, the elements of matrix \( Q \) can be written in terms of the element of matrix \( H \) from matrix theory and not be given for short.

2.4 The input power flow of the coupled system

For a structure excited by an external force, the magnitudes of vibration and sound radiation depend largely on the input power flow into the structure. Thus, it is important to study its characteristics of the input power flow.

According to the definition of the input power flow\(^{(1)}\), when a cosine harmonic line force is applied to the shell wall radially, the radial response of the shell wall at \( x=0 \) can be obtained from Eq.(20). Then, the total input power flow from this driving force is defined as follows

\[
P_{\text{input}} = \int_0^{2\pi} \frac{1}{2} \Re \left[ F_0 \cos(\omega t) \frac{\partial w(0) \ast}{\partial \mu} \right] R d\theta
\]

\[
= \pi \omega F_0 \xi_n \sqrt{\frac{E}{\rho_0 \left( 1 - \mu^2 \right)}} \int_{-\infty}^{\infty} \Im \left[ w(0) \ast \right] d(k_j R) \tag{21}
\]

where * denotes the complex conjugate, and

\[
\xi_n =\begin{cases} 2 & n=0 \\ 1 & n \neq 0 \end{cases} \tag{22}
\]

The non-dimensional power flow is defined as

\[
P'_{\text{input}} = \frac{P_{\text{input}}}{F_0 \pi} \sqrt{\rho_0 E R^2 \left( 1 - \mu^2 \right)} \tag{23}
\]
3 Numerical computation and result discussion

In this paper, an integrated numerical method discussed in Ref.[10] is employed to calculate the integral in Eq.(21). This method is to integrate numerically along the pure imaginary axis of the complex wave number domain in order to avoid singularities in the integrand function along the integration path. Structural damping is introduced into the shell material by modifying the Young’s modules $E$ to be complex as $E’ = E (1 - i\eta)$. Here, $\eta$ is damping factor.

Numerical computations are carried out subsequently and the input power flow from a cosine harmonic line force into a periodically ring-stiffened cylindrical shell submerged in fluid is studied. The following parameters of the coupled system have been used in the computations. The shell and the stiffener adopt the same material with $E = 1.92 \times 10^{11} \text{N/m}^2$, $\rho_s = 7850 \text{kg/m}^3$, $\mu = 0.3$ and $\gamma = 0.02$. The thickness-radius ratio of the shell is $h/R = 0.02$. The fluid parameters are $\rho_f = 1000 \text{kg/m}^3$ and $c_f = 1500 \text{m/s}$. The stiffener has a rectangular cross-section with width $b=2h$ and height $d=2h$, and the stiffener spacing is $L = 0.4R$. The amplitude of radial harmonic line pressure is supposed to be $F_0 = 1 \text{N/m}$. Fig.2 shows the non-dimensional input power flow $P_{\text{input}}$ into a ring-stiffened shell and a non-stiffened shell versus the non-dimensional frequency $\Omega$ for circumferential mode order $n = 1$, without considering the effect of hydrostatic pressure.

In some frequency bands, such as $0.68 < \Omega < 0.76$ and $1.3 < \Omega < 1.6$, the power flow input into the stiffened shell is much less than that into a shell without stiffeners. These frequency bands are named non-propagating bands, and the other frequency bands are called propagating bands in which the input power flow into the ring-stiffened shell is close to (or even exceed) that into the shell without stiffeners. It can be seen that the stiffeners greatly influence the input power flow in non-propagating bands. At $\Omega = 0.78$ and $\Omega = 1.34$, the power flow into the stiffened shell achieves its maximum, and is much more than that into an unstiffened shell. In propagating bands, the difference between the results with stiffeners and those without stiffeners is large, too.

Fig.3 shows the effect of hydrostatic pressure on input power flow into a ring-stiffened shell for circumferential mode $n = 0$, 1, and 5. There also exists propagating bands and non-propagating bands for a stiffened shell considering the effect of external hydrostatic pressure. For $n = 0$, the frequency bands of $0.70 < \Omega < 0.76$ and $1.3 < \Omega < 1.6$ are non-propagating bands, and the other frequency bands are propagating bands. The non-propagating bands for $n = 5$ are the
frequency bands of $0.5<\omega<0.72$ and $1.4<\omega<1.6$. As shown in these drawings, the curves of input power flow are shifted to left along the frequency axis mainly in non-propagating bands. It is because that the external hydrostatic pressure decreases the stiffness of the shell. Consequently, its natural frequency decreases due to the external hydrostatic pressure. The influence of hydrostatic pressure with $p_0=1\,\text{MPa}$ on the power flow is very small for low circumferential mode $n=0$ and 1. However, this influence for $n=5$ becomes larger. When the external hydrostatic pressure increases to $p_0=3\,\text{MPa}$, the influence of hydrostatic pressure on the power flow is larger especially for the high mode $n=5$.

Since the hydrostatic pressure shifts the curves of input power flow into a submerged ring-stiffened shell, the peaks and valleys of the power flow are shifted. It is significant for the vibration and noise of control.

![Graphs showing the effect of hydrostatic pressure on input power flow into a ring-stiffened shell.](image)

Fig. 3 Effect of hydrostatic pressure on input power flow into a ring-stiffened shell

(a) $n=0$; (b) $n=1$; (c) $n=5$; — $p_0=0$, ··· $p_0=1\,\text{MPa}$, ··· $p_0=3\,\text{MPa}$

4 Conclusions

By adopting a periodic structure theory, a periodically ring-stiffened cylindrical submerged shell has been investigated. The input power flow for this coupled system induced by a cosine harmonic circumferential line force has been obtained and the effect of external hydrostatic
There are propagating bands and non-propagating bands for a stiffened shell. The high external hydrostatic pressure influences the input power flow into the coupled system mainly in non-propagating bands. The external pressure will shift the curves of power flow to left along the frequency axis. With increasing the circumferential mode $n$ and the external hydrostatic pressure $p_0$, the effect of hydrostatic pressure on input power flow is more obvious. Hydrostatic pressure of a relative high value cannot be neglected when analyzing the vibration and noise of a submerged stiffened shell.

References


静水压力对水下环肋圆柱壳输入功率流的影响

刘志忠，李天匀，朱翔，张俊杰

（华中科技大学船舶学院，武汉430074）

摘要：研究了考虑静水压力时水下环肋圆柱壳在周向余弦线分布力激励下的输入功率流特性。圆柱壳体和外流场的振动分别由 Flugge 壳体方程和 Helmholtz 波动方程描述，静水压力的影响以额外应力的形式计入壳体振动方程当中。探讨了静水压力对输入功率流的影响，结果表明外部静水压力使输入功率流曲线沿频率轴往左移动，即往低频方向移动。静水压力越大，影响越大，周向模态数越大，影响越大。文中结果对水下环肋圆柱壳的振动与噪声控制有一定的指导意义。

关键词：静水压力；输入功率流；环肋圆柱壳；水下圆柱壳

中图分类号：O328 文献标识码：A

作者简介：刘志忠（1980-），男，华中科技大学船舶学院博士研究生；
李天匀（1969-），男，华中科技大学船舶学院教授，博士生导师；
朱翔（1980-），男，博士，华中科技大学船舶学院讲师；
张俊杰（1982-），男，华中科技大学船舶学院博士研究生。
Study on Operational Modal Parameters Identification of Ship Structures

JIANG Da-zheng, HONG Ming, ZHOU Li
(School of Naval Architecture Engineering, Faculty of Vehicle Engineering and Mechanics, State Key Laboratory of Structural Analysis for Industrial Equipment, Dalian University of Technology, Dalian 116024, China)

Abstract: Operational modal analysis is a procedure which allows extracting modal parameters from a structure only based on response data under ambient excitation. It is based on the assumption that input to the structure is stationary white noise. In this paper, the application feasibility of operational modal analysis on ship structures is discussed. And band stop filter is proposed to filter the obvious harmonic energy to improve the identification precision of real modes. Then the auto-cross spectrum density method is discussed and verified by a simply supported beam experiment with white noise and harmonic excitations. This method is testified to be effective to identify the natural frequencies and operating deflection shapes. So it feasible and significant to identify modal parameters of ship structures under ambient excitations.

Key words: operational modal analysis; ship structures; parameter identification; band stop filter

CLC number: U661.4 Document code: A

1 Introduction

Modal analysis method includes experimental modal analysis (EMA) method and operational modal analysis (OMA). Operational modal analysis procedure is an efficient technique to identify modal properties of structures only based on response data during operation. Comparing with traditional modal analysis methods, OMA has some typical characteristics as follows:

(1) OMA can extract modal parameters of large constructions only from response data under ambient excitations without additional excitations. This is a large improvement to traditional methods which can not identify modal properties efficiently because of failing to apply effective excitations.

(2) The identified results with OMA can reflect the real dynamic characteristic of the structure during operation. It can be used to test the changes of modal parameters caused by ambient excitation and the structure health monitoring and damage diagnosis.
(3) This method only depends on ambient excitations without extra force, which is convenient and economical to large constructions such as bridge, ocean engineering and high-rise buildings, etc. It can not only save the expense of transport and setting but also avoid possible damnification to the structure.

The conventional modal analysis to a ship requires it to berth at a wharf and identifies the modal properties through special excitation devices and in special situation. In practice however, this method is not only difficult to realize but also unable to obtain satisfied effect. With the increase of dimension and tonnage of modern ships, to design and produce an excitation device which can excite the resonance of the whole ship with enough energy is difficult and not economical. At the same time, the excitations to an operating ship are complicated, the magnitude and position of the excitations can not be identified exactly. If OMA can be used to analyze modal parameters of a certain vibration area of a ship, it can not only give verification to the theory analysis, but also help to propose reasonable solution to the structure including unacceptable vibration. So study on operational modal parameters identification of ship structures based on ambient excitations is significant and valuable.

In this paper, the excitations on an operating ship are discussed and assumed to be the combination of white noise and harmonic excitations. The frequency domain method of digital filtering is proposed to filter the harmonic force priorly, then with the filtered data the auto-cross spectrum density method\cite{3} is discussed to calculate modal parameters. This method is verified with a simple supported beam experiment under white noise and a harmonic excitation. The experiment results are acceptable which verify the proposed filtrated-OMA method is effective to identify parameters of ship structures under ambient excitations.

2 The excitations on an operational ship

At present, OMA is limited to the case when the excitations can be assumed to white noise inputs. In practice however, an operational ship is vibrating due to many obvious harmonic excitations in addition to stationary white noise. Generally the excitations are mainly produced by random wave, wind, main engine and the propeller, etc. And the frequencies of obvious harmonic loads are usually known priorly, which usually includes the frequencies of main engine and propeller shaft, blade frequency and double blade frequency of propeller. Fig.1 shows a typical frequency spectrum of a ship mast, it is noted that the blade frequency (21.1Hz) and double blade frequency (42.2Hz) are obvious or dominating, the magnitude of double blade frequency is larger than the other modes, which possibly affects the identification of the real mode. If the obvious harmonic excitations can be restrained or filtered, the identification precision can be improved. So the excitations to an operational ship can be assumed to be white noise including harmonic excitations and the frequencies of harmonic force are known priorly.
3 Theoretical background

The band stop filter technique is discussed and used to filter the known harmonic force. Then the auto-cross spectrum density method is introduced to identify the operational modal parameters with the filtered data.

3.1 The frequency domain method of digital filtering

The main functions of digital filtering are filtrating the noise or false element from the testing signal, improving the signal to noise ratio (S/N), smoothing analytical data, suppressing interference signals and separating frequency components, etc. The frequency domain method of digital filtering transforms the input sampling data with FFT algorithm, and then analyzes its frequency spectrum. The frequencies to be filtered are set to be zero or do this after adding a transition frequency zone, such as a transition zone of cosine window function can be added between the passband and the stopband. At last the time domain data can be obtained by doing discrete inverse Fourier transform to the filtrated data.

The equation of the frequency domain method of digital filtering can be written as

$$y(r) = \sum_{k=0}^{N-1} H(k) X(k)e^{j2\pi rk/N}$$

where $X$ is the discrete Fourier transform of input signal $x$; $H$ is the frequency response function of the filter, which can determine the mode and character of filtering.

It is assumed that $f_u$ is upper cut-off frequency, $f_d$ is lower cut-off frequency, $\Delta f$ is frequency resolution. In ideal condition, the FRF of band stop filter which is used in this paper can be written as

$$H(k) = \begin{cases} 1 & (k \Delta f \leq f_d, k \Delta f \geq f_u) \\ 0 & \text{(other)} \end{cases}$$

The frequency method of digital filtering is easy and convenient to calculate and has a high control accuracy of filtering frequency band. Frequency domain method can choose frequency freely and calculate faster than the equivalent time domain convolution method because the
Fourier spectrum of signal has a simple multiplication relation with the frequency character of the filter. Also time shift will not occur like time domain filtering method. So the band stop filter will be used to filter an obvious harmonic frequencies band, and then the filtered time domain data can be used to identify modal parameters.

3.2 The auto-cross spectrum density method

Auto-cross spectrum density method is a simple and convenient procedure to identify modal parameters based on ambient excitations. It originated from the Peak-Picking method[6]. To the vibration based on ambient excitation, the FRF will be meaningless, so the FRF is substituted with the auto power spectrum and cross power spectrum density.

When the excitation force can be obtained, the force signal of excitation point is usually used as input and the response signals of the structure as output. To a real mode system, the FRF can be obtained from the relationship between the excitation and the response as

$$h_{ik}(\omega) = \frac{x_i(\omega)}{f_k(\omega)} = \sum_{r=1}^{N} \frac{\phi_{ir} \phi_{kr}}{k \omega + m_i + j \omega c_i} = \sum_{r=1}^{N} \frac{\phi_{ir} \phi_{kr}}{(j \omega - \lambda_r) (j \omega - \lambda^*_r)}$$

where $N$ is the mode number. $\phi_{ir}$ and $\phi_{kr}$ are the vector of mode shape of the $r$th mode shape in point $i$ and point $k$ respectively. $\lambda_r$ and $\lambda^*_r$ are a pair of con-eigenvalue. It is noted that the FRF contains all the modal information of the structure.

When the excitation force of the structure cannot be measured while the response is available, the response of a certain reference point can be assumed to be input (motion excitation), the responses of the other points have a certain linear relativity with the reference point, then the transfer function between the response points and the reference point can be obtained to do system identification. Defining a stationary reference point $P$, so the transmissibility can be written as

$$\alpha_i(\omega) = \frac{x_i(\omega)}{X_i(\omega)}$$

To some meaningful eigenfrequency $\omega_i$, $\alpha_i(\omega_i)$ can be seemed as the structure’s operational mode with relevant frequency. The dynamic displacement response $x_i(\omega)$ of point $i$ in structure can be expressed with the excitation force $f_k(\omega)$ of point $k$ and the transfer function $h_{ik}$ of the system can be written as:

$$x_i(\omega) = \sum_{k=1}^{m} h_{ik}(\omega) f_k(\omega)$$

If the signal of the excitation force is flat spectrum, the spectrum density function is approximate uniform distribution in the whole mode frequency range, so the excitation force in the structure satisfies $f_k(\omega) = f(\omega) = C_1$, where $C_1$ is a constant. Then Eq.(5) can be written as

$$x_i(\omega) = f(\omega) \sum_{k=1}^{m} h_{ik}(\omega) = C_1 h_i(\omega)$$

where $h_i(\omega)$ is the lumped frequency response function.
Eq. (6) shows that \( x_i(\omega) \) and \( h_i(\omega) \) (real mode) are equivalent, so the eigenfrequency can be obtained from the response spectrum \( x_i(\omega) \) of the structure directly. By the substitution of Eq. (6) into Eq. (4), then Eq. (4) can be formulated:

\[
\alpha_i(\omega) = \frac{f(\omega) \sum_{k=1}^{m} h_{ik}(\omega)}{\sum_{k=1}^{m} h_{ik}(\omega)} = \frac{\sum_{k=1}^{m} h_{ik}(\omega)}{\sum_{k=1}^{m} h_{ik}(\omega)}(7)
\]

By comparing Eq. (7) with Eq. (3), it can be concluded that transmissibility is relative to the mode parameters directly. It is assumed that the real mode can be separated effectively. So the system response at \( \omega_r = |\lambda_r| \) is dominated mainly by the vibration of the \( r \)th mode, the contribution of the other orders can be ignored, then Eq. (7) can be written as

\[
\alpha_i(\omega) = \frac{\sum_{k=1}^{m} h_{ik}(\omega)}{\sum_{k=1}^{m} h_{ipk}(\omega)} = \frac{\phi_{ir}}{(j\omega - \lambda_r)(j\omega - \lambda_r)} \sum_{k=1}^{m} \phi_{ikr} = \phi_{ir} \phi_{pr} \phi_{ir}(8)
\]

where \( p \) is assumed to be the reference point. So \( \phi_{pr} \) is constant to an eigenfrequency. Then Eq. (8) can be written as

\[
\alpha_i(\omega) = \frac{\phi_{ir}}{\phi_{pr}} = C_2 \phi_{ir} \phi_{ir}(9)
\]

where \( C_2 \) is a constant. It is noted that when the frequency is \( \omega_r \), the structure’s ODS (Operating deflection shapes) can be obtained by reading the value (amplitude and phase) of \( \alpha_i(\omega) \) in the location of \( \omega_r \), it can be approximately seemed as the \( r \)th mode shape of the structure.

3.3 The identification procedure

This section briefly introduces the identification procedure of the proposed method as follows:

1. Band stop filter. After obtaining the response data, it is transformed by FFT algorithm. If the signals contain some obvious harmonic elements and their frequencies are known priorly, it will be filtered by a band stop filter. Then the filtered time domain data is obtained by using IFFT fast algorithm.

2. Choose a reference point. The selection of the reference point has a great influence to the identification results. The point should have significant response in every eigenfrequency and it is not close to the node of concerned mode.

3. Calculate the auto power spectrums of the response points, the amplitude and phase of cross power spectrum between the response points and the reference point, the coherence function and transmissibility, then draw their figure respectively.

4. Identify the natural frequencies. It is verified that the auto power spectrums of response points or cross power spectrum between the response points and the reference point can
substitute the amplitude of FRF, and they have the same frequency in every peak. So the natural frequencies can be obtained by reading auto and cross power spectrums. To reject false peaks, it is essential to use the phase of cross power spectrum and coherence to testify these peaks. It is proved by many practical applications that the peak can be concluded to be natural frequency point if its phase of cross power spectrum is close to 0° or 180° (±30°), and also the coherence in this point is above 0.95. Then the natural frequencies can be obtained.

(5) Calculate damping ratio. The damping ratio of every frequency can be calculated by half-power bandwidth method.

(6) Calculate mode shapes. From the above theory, it is noted that the transmissibility in every eigenfrequency can substitute mode shape approximately, and then the magnitude of mode can be read from the amplitude of transmissibility, the direction of mode can be obtained from the phase of cross power spectrum or the symbol of the real part of transmissibility. It is noted that the mode shape with this method is actually an operational deflection curve but not the real mode shape.

4 A simply supported beam experiment

This experiment utilized a simply supported beam\(^7\) (700mm × 50mm × 10mm) to simulate a ship structure under white noise and obvious harmonic excitations, and identify the modal parameters of the structure to testify the validity of the proposed method. The experimental devices and settings are showed in the Fig.2. Input forces were applied through two shakers which can provide stationary noise excitation (shaker 2) and harmonic excitation (shaker 1), respectively. Brüel & Kjær® generator module was used to generate white noise signal and then input it to signal amplifier 2 to drive shaker 2. Signal generator 1 generates harmonic signal (71.7Hz) to drive shaker 1. The acceleration responses were obtained by a 4-channels data acquisition system and channel 1 was chosen to be the reference channel. Sampling of the signals was done at 1500Hz which is good to compute frequencies up to 750Hz. The length of FFT was set as 2048.

![Fig.2 The experimental devices and settings](image)
Fig. 3 shows the frequency spectrums of responses from four channels, it is noted that the inputs excite four modes of the beam obviously, but they have different amplitudes. This may be caused by the different positions of the two shakers and four sensors. Also the amplitude of the known harmonic excitation (71.7Hz) is obvious and larger than most of the natural modes, this may affect and disturb the identification of the real mode. So this harmonic energy was filtered through band stop filter. The frequency band 68-75Hz was chosen to be filtered. Fig.4 shows the filtering procedure and result of channel 2. Pictures (1) and (3) show the original and filtered signals in time domain respectively and pictures (2) and (4) are their spectra. It is noted that the real modes are more obvious after filtering. Then the filtered data in time domain can be used to calculate the modal parameters.
The auto power and cross power spectrum density are calculated with Welch method\cite{4} which is a modified average periodogram algorithm. Fig.5 shows the auto power spectrum density of original signals and filtered signals. It is noted that the four modes calculated with filtered data are more visible. Then four natural frequencies can be roughly obtained by comparing with the spectra of four channels and the amplitude of cross power spectrum density which are 39.55Hz, 149.41Hz, 331.79Hz and 577.88Hz, respectively. The amplitudes and phases of cross power spectrum between the reference channel (channel 1) and response channels (channels 2-4) are showed in Fig.6 and Fig.7 respectively. Fig.8 shows the coherence between reference channel and response channels. Comparing with the values of the coherence and the phases of cross power spectrum in the above eigenfrequencies, it is noted that their coherences are all larger than 0.95 and the phases are all around 0° or 180°, so the four natural frequencies can be identified finally.

![Fig.5 Auto power spectrum density of original signal and filtered signals](image)

The damping ratios at eigenfrequencies can be obtained by half-power bandwidth method\cite{8} from the curves of power spectrum densities. And the calculation results are showed in the Tab.1. It is noted that the calculated natural frequencies are all larger than the FEM results, this occurs because the two shakers all withstand the beam underneath which may enhance the stiffness of the system which results in the increase of frequencies. So the calculated natural frequencies are credible and acceptable. The damping ratios have significant errors compared with experimental results, so the identification of damping with this method is not precise. But the damping is not what we most concerned in parameters identification of ship structures and vibration control.
Fig. 7 The phases of cross power spectrum densities

Fig. 8 The coherence
Tab.1 Calculation results and comparison

<table>
<thead>
<tr>
<th>Mode</th>
<th>Frequency (Hz)</th>
<th>Error (%)</th>
<th>Damping (%)</th>
<th>Error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>A</td>
<td>B</td>
<td>A</td>
<td>C</td>
</tr>
<tr>
<td>1</td>
<td>39.55</td>
<td>37.62</td>
<td>4.9</td>
<td>5.9</td>
</tr>
<tr>
<td>2</td>
<td>149.41</td>
<td>142.50</td>
<td>4.6</td>
<td>0.98</td>
</tr>
<tr>
<td>3</td>
<td>331.79</td>
<td>320.70</td>
<td>3.3</td>
<td>0.53</td>
</tr>
<tr>
<td>4</td>
<td>577.88</td>
<td>570.33</td>
<td>1.3</td>
<td>0.45</td>
</tr>
</tbody>
</table>

Note: Frequency A and damping A are obtained by the proposed method. Frequency B is calculated by finite element method. Damping C is obtained by experimental modal analysis method.

Fig.9 is the transmissibility between channels 1 and 2, 3 and 4. The values and directions of the mode shapes can be respectively obtained from the amplitudes of transmissibility and the phases of cross power spectrum densities. Then the mode shapes can be drawn roughly as shown in Fig.10.

![Fig.9 The transmissibility](image1)

(1) Channels 1 and 2

![Fig.10 The mode shapes](image2)

(2) Channels 1 and 3

(3) Channels 1 and 4

In the process of this experiment, different reference points were chosen to verify the results, but only one typical situation is discussed in this paper due to limited space. It is worth to notice that the selection of the reference point location is not random, which is directly related to the identification precision. It is known that the node of every mode has no response when the structure is resonating under this mode. Also if the response of one point in the structure is
small, the noise will occupy a large proportion of the response signal, which can lower the signal to noise ratio (S/N) correspondingly and affect the identification. So the preferable reference point should have significant response and not very closed to the node in concerned mode.

It can be concluded that the proposed method is efficient to identify the natural frequencies to the system of sparse mode and low damping, and can also identify the operational deflection curve roughly. The error of damping is the largest. The experiment verifies the effectiveness and feasibility of the discussed filtered-OMA method with the assumption of white noise including harmonic excitations\[9\]. Then this method can be used to analyze a real ship structure further.

5 Conclusions

In this paper, the feasibility and significance of operational modal analysis used to analyze the vibration of operating ship structures were discussed first. Then the excitations to an operating ship were discussed and assumed to be white noise including harmonic excitations.

Band stop filter was proposed to filter the obvious harmonic excitation on improve the identification precision. Then the auto-cross spectrum density method was introduced to identify modal parameters from the filtered response signals.

From the experimental results of a simply supported beam, it was concluded that the proposed filtered-OMA method is a simple and convenient method to identify modal parameters based on ambient excitations which include white noise and harmonic excitations. It is fairly accurate to calculate the eigenfrequencies and the operating deflection shapes which can be seemed as the mode shapes appreciatively.

As mentioned in the end of section 4, attention should be paid to the selection of reference point to increase the S/N and the identification precision.

It is conclude that the proposed method is effective and feasible to the unacceptable vibration measure and modification during ship voyage. It can calculate the modal parameters conveniently and help engineers to propose reasonable solution to control or reduce the unacceptable vibration. The experiments to an operating ship mast and compass deck have been done recently, and more study and researches are carrying out by our research team.

References

船舶结构运行模态参数辨识研究

姜大正，洪 明，周 力

（大连理工大学工业装备结构分析国家重点实验室 运载工程与力学学部 船舶工程学院，辽宁 大连 116024）

摘要：运行模态分析是仅基于结构运行状态下的响应来提取结构模态参数的一种方法，通常假设环境激励为白噪声进行分析。实际上，船舶运行过程所受激励十分复杂，不能简单地假设为白噪声激励，由主机和螺旋桨等产生的确定频率激励同样存在，因此，航行中船舶所受激励可以假设为白噪声和简谐激励同时存在。进一步讨论了运行模态技术在船舶结构模态分析中应用的可行性和前景。研究表明使用带阻数字滤波技术滤除激励中的简谐成分来提高结构真实模态的辨识精度。讨论了自互功率谱密度法，并研究使用此方法提取同时受白噪声和简谐激励作用的简支梁的模态参数。实验结果表明该方法能够较准确地识别出同时受白噪声和简谐激励的结构模态参数。这对于分析航行中船舶有害振动结构的模态数据，并实施有效的减振优化措施具有很广泛的实际意义。

关键词：运行模态分析；船舶结构；模态参数；带阻滤波

中图分类号：U661.4 文献标识码：A

作者简介：姜大正（1984-），男，大连理工大学船舶工程学院硕士研究生；
洪 明（1959-），男，大连理工大学船舶工程学院教授；
周 力（1965-），女，大连理工大学船舶工程学院副教授。
《船舶力学》第四届编辑委员会名单
(EDITORIAL COMMITTEE OF JOURNAL OF SHIP MECHANICS OF THE 4TH TERM)

主任委员 Chairman: 崔维成 Cui Weicheng

副主任委员 Vice Chairman: 颜开 Yan Kai 万正权 Wan Zhengquan 俞孟萨 Yu Mengsa

编委 Members (以姓氏首字母为序):

主任委员 Chairman: 崔维成 Cui Weicheng

副主任委员 Vice Chairman: 颜开 Yan Kai 万正权 Wan Zhengquan 俞孟萨 Yu Mengsa

编委 Members (以姓氏首字母为序):

主编 Chief Editor: 沈泓萃 Shen Hongcui

常务副总编辑 Executive Associate Chief Editor: 黄国荣 Huang Guorong

船舶力学 CHUANBO LIXUE
(月刊·1997 年创刊)
第 15 卷第 3 期 2011 年 3 月出版

ISSN 1007-7294 CN32-1468/U

定价: 18.00 元

PDF 文件使用 "pdfFactory Pro" 试用版本创建 www.fineprint.cn
Contents

Hydrodynamics

Numerical simulation on the process of supercavity development and the planing state of supercavinating vehicle................................. ZHOU Jing-jun YU Kai-ping ZHANG Guang (199)

Level set method for simulation of cavitating flows........................................ HUANG Biao WANG Guo-yu ZHANG Min-di CHEN Wei-qing (207)

Optimal thrust allocation based on fuel-efficiency for dynamic positioning system................................................................. YANG Shi-zhi WANG Lei ZHANG Shen (217)

Investigation on sloshing effects of tank liquid on the FLNG vessel responses in frequency domain......................................................... ZHAO Wen-hua HU Zhi-qiang YANG Jian-min WEI Yu-feng (227)

Performance comparisons of vortex-induced vibration suppression devices for top tensioned riser...................................................... LI Xiao-ming GUO Hai-yan ZHANG Yong-bo LIU Xiao-chun (238)

Vortex-induced vibration analysis of steel catenary riser..................................................................................................................... RAO Zhi-biao FU Shi-xiao YANG Jian-min (245)

Structural mechanics

Influence of liquid cargo in tank on crashworthiness of double-skin side structure................................................................. CUI Jie ZHANG A-man YAO Xiong-liang YANG Shu-tao (259)

Comparative studies of the transverse structure design wave loads for a trimaran by model tests and rule calculations.......................... WANG Xue-liang HU Jia-jun GU Xue-kang GENG Yan-chao XU Chun (269)

A comparison of different rules for the spherical pressure hull of deep manned submersibles.............................................................. PAN Bin-bin CUI Wei-cheng (276)

On implementing a practical algorithm to generate fatigue loading history or spectrum from short time measurement................. LI Shan-shan HUANG Zhi-bo CUI Wei-cheng (286)

Effect of hydrostatic pressure on input power flow in submerged ring-stiffened cylindrical shells............................................................ LIU Zhi-zhong LI Tian-yu ZHU Xiang ZHANG Jun-jie (301)

Study on operational modal parameters identification of ship structures...................................................................................... JIANG Da-zheng HONG Ming ZHOU Li (313)

E-mail: huanggr@cssrc.com.cn  网址:  http://www.cssrc.com.cn
目录

流体力学
超空泡发展过程研究以及对两种研究潜行力方法的评估………………周景军 于开平 张 广 (199)
Level set 方法在空化流动计算中的应用……………………………黄 彪 王国玉 张敏弟 陈维清 (207)
基于能耗最优化的动力定位推力分配逻辑算法研究…………………李世民 王 磊 张 申 (217)
频域范围内液舱晃荡对 FLNG 运动影响的研究………………………赵文华 胡志强 杨建民 魏跃峰 (227)
顶张力立管涡激振动抑制装置性能比较……………………………李效民 郭海燕 张永波 刘晓春 (238)
钢悬链线的涡激振动分析………………………………………………饶志标 吴世超 杨建民 (245)

结构力学
舱内液货对双层舷侧碰撞性能的影响分析…………………………崔 杰 张阿漫 姚熊亮 杨树涛 (259)
三体船横向结构波浪设计载荷与规范比较研究…………………………汪雪良 胡嘉骏 顾学康 耿彦超 徐 航 (269)
深潜器载人球壳规范设计公式的比较……………………………………潘彬彬 崔维成 (276)
从短期测量数据来生成疲劳载荷的一种实用算法的开发………………李珊珊 黄志博 崔维成 (286)


船舶力学

第 15 卷 第 3 期 2011 年 03 月