Numerical simulation on fluid-structure coupling under impact loads between rotating deformable body and rigid body by FLUENT and LS-DYNA

Bincheng Huang [1, a], Baotai Yao [2, b] and Han Li [3, c]

[1]School of Transportation Science & Engineering, Beihang University, Beijing 100191, China

[2]713th Research Institute of China Shipbuilding Industry Corporation, Zhengzhou 450015, China

[3]Patent Examination Cooperation Center of the Patent Office, SIPO, Guangzhou 510530, China

[a]huangbc1987@126.com, [b]yaobaotai@126.com, [c]leonchains@gmail.com

Abstract. The fluid-structure coupling phenomena increase the difficulty in designing a safe underwater structure, especially for those structures subject to impact loads. Although numerous experimental researches on fluid-structure coupling have focused on vibration of rigid body under water around the balance position, little previous work considers the elastic or large plastic deformation of structures driven by fluid. In the present paper, an experiment platform for investigating on the dynamic response and verify the simulation algorithm for fluid-structure coupling problem, with both large displacement and large deformation, was set up. Numerical simulation involved collision process of deformable structures driven by fluid is engaged, and several interface coupling parameters were discussed. And the distributions of pressure and flow velocity in fluid domain, as well as the strain distribution in structure domain, were also obtained. The comparison between numerical simulation and experimental measurement are acceptable for engineering problems.

Keywords: Deformation structure driven by fluid, Fluid-structure coupling, Experiment platform explored, Multi-physics simulation.

1. Introduction

Phenomenon of fluid-structure coupling has been widely existed in whole-vertical or parts manufacturing process. Not only effect on the making precision but also driving safety, especially would induced the losses which involved staff and property. To date, researches of fluid-structure coupling which generated from two different fields of solid mechanics and fluid mechanics are still deficiency, thus scientific explanation and guiding become more significant for studying above problems. And most of researches on fluid-structure coupling focus on the deformation and displacement due to their essentiality and practical meaning[1,2].Coupling interface usually divided into two groups: One is only generated from interface between fluid and solid domain, would not influenced on constitutive equation such as pneumatic.And the other problem is that parts or all from fluid zone and solid zone are overlapped such as percolation.This paper mainly aiming at the first kind to research[3].

In this paper, simulation analysis of Fluid-driven deformable Structure in crashing process is mainly discussed by using Fluent and LS-DYNA. In order to enhance the calculation accuracy of each physical field, Simulation method and data transmission mode are involved. Research of fluid-structure coupling generally depend on wind tunnel test, however, for provide trade-off between cost and accuracy, new experiment platform which can be used as investigating on rotational impact process of shell with fluid driven is developed. The high cost of the experiment, physical testing and also the development of new finite element codes make design validation by means of numerical methods very attractive.

2. Fluid-structure Coupling Control Equation

Work characteristic of fluid-structure coupling shows coupling effect which take place from the interface between two phases, control equations of structure and fluid domain are established through some conditions such as displacement coordination, equilibrium and energy conservation [4].

Fluid domain equation combined with structure domain equation, and also should meet following equations: mass conservation equation in fluid domain (Eq.1), momentum conservation equation in fluid domain(Eq.2), elastic structure dynamics equation(Eq.3).

$$\nabla \boldsymbol{u} = 0 \tag{1}$$

$$\rho \frac{Du}{Dt} = \rho F - \nabla p + \mu \nabla u^2 \tag{2}$$

$$\rho_s \frac{\partial^2 l}{\partial t^2} + \xi(\frac{\partial l}{\partial t}) - \nabla \cdot \sigma = G \tag{3}$$

In the equations, ∇ represents operator, u ,the fluid velocity, ρ is fluid density, t represents time, p being pressure in fluid domain, μ is viscosity coefficient in fluid domain, F ,the volume force in fluid domain, l is structure body displacement, ρ_s is structure body density, σ is damping coefficient, ξ is stress tensor, and G is volume force of structure body.

There are some coupling relationships between each parameter in the established process of combining fluid domain and structure domain constitutive equations. And data transfer also follow the interface condition of coupling [5].

Though comparative analysis of parallel coupling and sequence coupling, find that implementation for parallel algorithm is not realized easily in practical application due to requirement for a whole solvers with fluid and structure. By contrast, sequence coupling algorithm only need to choose the suitable step size, it could calculated more real simulation results. From above all, sequence coupling algorithm is applied in this paper to research the vibration process of fluid-driven deformable Structure under water. Then, the problem is divided into two groups: rotation process of solid boundary which is calculated by the Fluent, rotation impact process with LS-DYNA. The calculation results from user-defined programs are loaded on LS-DYNA which are served as initial condition in collision process.

3. Experiments

Most of the research on fluid-structure coupling experiment has focused on vibration or rigid body displacement at the balance position. Relatively little attention has been given to their elastic deformation and structure large displacement, especially the deformation motion of fluid driven structure [6-7]. Therefore, a relative simple experiment equipment on structure and principle is designed for researching coupling phenomenon of structure large displacement and deformation.

3.1 Experiments Platform

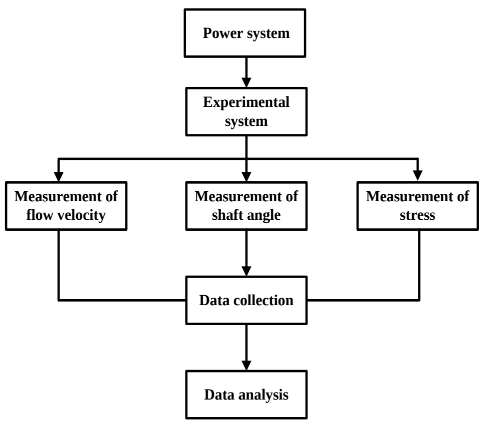


Fig.1. Measurement principle of experimental platform [8]

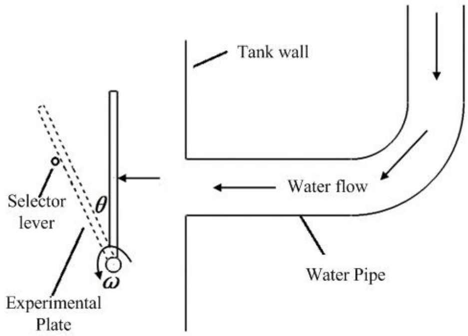


Fig.2. Schematic diagram of structure rotation driven by fluids [8]

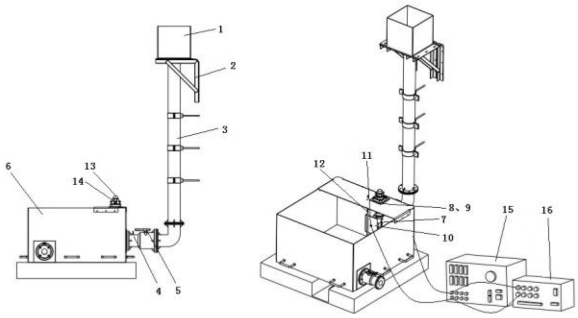


Fig.3. Full view of fluid-structure coupling experimental device [8]

1-Storage tank;2- Support;3-PVC Pipe;4- Target flowmeter;5- valve;6- Water channel;7-Rotation shaft;8-Bearing;9- Bearing block;10- Baffle;11- Strain gauge;12-Collision selector lever;13- Angular displacement sensor;14- Flexible coupling;15- Dynamic strain indicator;16- Signal collection transaction and analysis apparatus;

In the equipment, fluid in the upper water storage tank flows into flume as shown in fig.2, and drives the shell to rotate. Inlet vertical to the initial plane of plate (fig.2) .Plate could rotated automatically in high dam plunge pool, and impacted on rigid crosspiece rod at the 40 degree where the largest deformation is appeared. For the enough small rotational inertia, lightweight material such as Nylon is used to manufacture shaft. And a pressure device fitted on water storage tank is designed to generate different kinds of flow velocity.

3.2 Test result

Displacement-Time history of shell on the four kinds of driven-flow speed is received by using angular displacement sensor, as shown in fig.3.Four curves from figures have the similar shapes show that impact process which driven-structure under fluid force impact on the selector lever after rotating 40 degrees. Meanwhile, more high speed the fluid has, larger angle the shell has, because of the great kinetic energy which is induced by inlet fluid velocity.

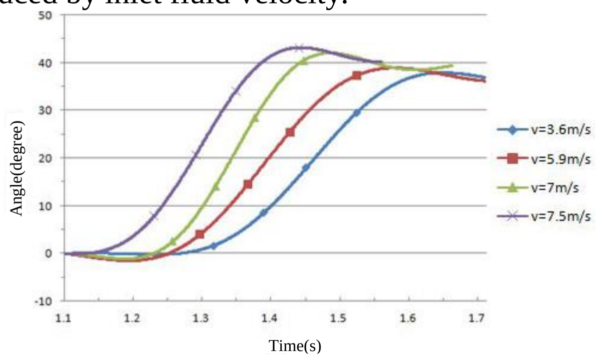


Fig.4. Time history of the plate rotating

In order to analysis the plate mechanical properties in the interaction process with fluid, Torque-Time history under different driven flow velocity is obtained as shown in fig.4.

Stress of installation position is measured by applying strain gauge which is fitted on the plate, the numbering is shown in fig.5. Number 1 and 3 are installed on the back face of plate, number 2 is installed on the plate front.

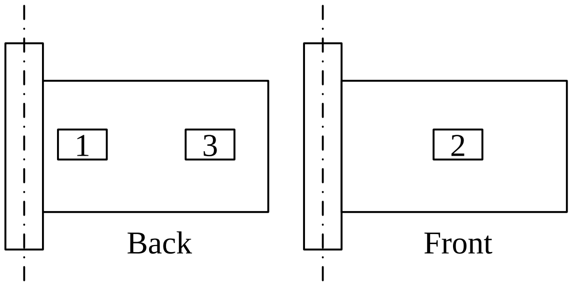


Fig.5. Adhesion positios of strain gauges

Maximum stress of measuring points in the rotating impact process as shown in table 1.

Table1. The maximum stress on three measured points at different impact speeds

Velocity	Position	Maximum stress(Mpa)
V= $3.6m/s$	Back 1	5.62
	Front2	0.58
	Back3	8.28
V= $5.9m/s$	Back 1	7.92
	Front2	0.72
	Back3	15.48
V= $7m/s$	Back 1	8.21
	Front2	0.73
	Back3	16.56

From table 1, it is clearly shown that maximum stress of each measuring points become greater which there is higher flow velocity.And the maximum stress on the impact position of plate and selector lever is greater than the rest of stress.

4. Numerical simulation

In the present research of bidirectional transient fluid-structure coupling, most of study objects just exist small deformation and displacement in the structure domain. Though Workbench could be used for solving fluid-structure coupling problems, it is not suitable for engaging in dynamic impact problems. Therefore, a simulation method which is combined FlUENT and LS-DYNA is proposed to investigate the rotational collision process of driven-structure, and data is transmitted by Matlab. Collision process could be divided into two parts: one is the solid boundary rotational process, the other is the impacting process, where pressure of fluid imposed on plate is abstracted by Matlab from Fluent and introduced into LS-DYNA.

4.1 Structure model of fluid domain and structure domain

Fluid domain model is defined by the actual situation in experimental condition, specific size is shown in table 2.The size and material of collision plate is the same as real condition. Length is 165mm, width is 90mm, height is 3mm, and material properties is shown in table 3. The diameter of selector lever is 10mm, material is 45# steel.

Table2. Fluid domain size in the tank

Parameter	Size/m
Length	0.9
Width	0.9
Height	0.4

Table3. Material attribution of collision plate

Attribution	Value
Density (kg/m^3)	2700
Young modulus (Pa)	2e11
Poisson ration	0.27

4.2 Simulation results

Pressure and flow velocity distribution of plate is obtained though Fluent simulation. The total time from rotating to cease is 0.054s, Pressure distributions of 0.014s and 0.054s are respectively obtained, as shown in fig.6 and fig.7.

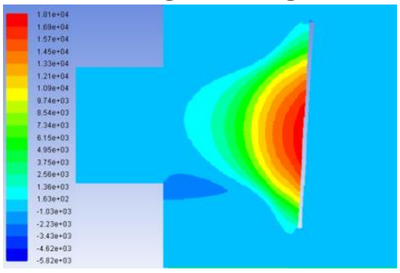


Fig.6 Pressure distribution in fluid domain at 0.014s

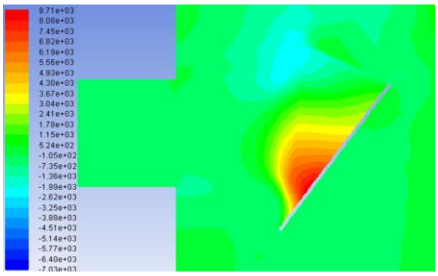


Fig.7 Pressure distribution in fluid domain at 0.054s

Upstream face is always under the barotropic state because of the impacting from water flow. It is also clearly that pressure difference between two sides of plate is observed. Boundary condition of fluid field and pressure distribution would been changed due to the plate position. Flow velocity in fluid domain at 0.014s and 0.054s is respectively obtained, as shown in fig.8 and fig.9.

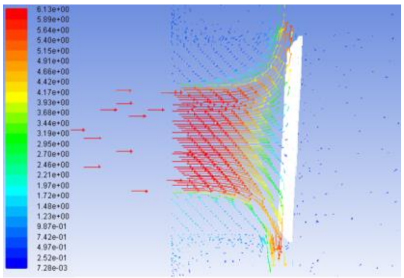


Fig.8 Diagram of velocity vector in fluid domain at 0.014s

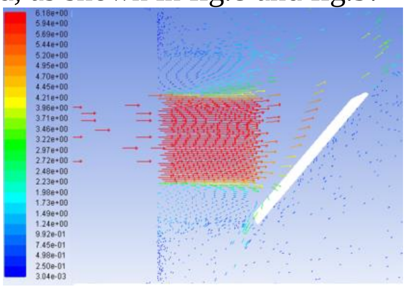


Fig.9 Diagram of velocity vector in fluid domain at 0.054s

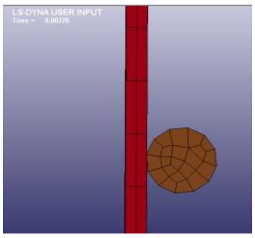


Fig.10 Finite element model at collision moment

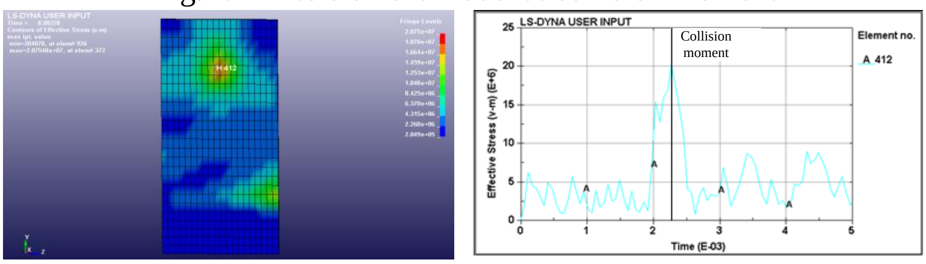


Fig.11 (a) Stress distribution at collision moment and (b) stress-time history at collision point

From the flow velocity distribution it is also possible to observe the trajectory process of plate in solid domain. When the plate is driven by impacting of flow which the flow midline at entry of circular tube as center, and velocity decreases from plate center to around. The angle between flow and plate surface is declined gradually while at certain angle of plate. Maximum flow velocity always at the circular tube inlet midline. Fluid-structure coupling parameters is pressure on the plate which is acted by flow, also is the input parameters of impact and collision process with LS-DYNA. Then, stress distribution of collision moment of plate and shaft could be obtained.

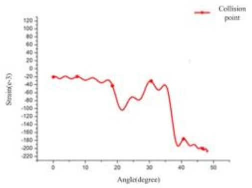
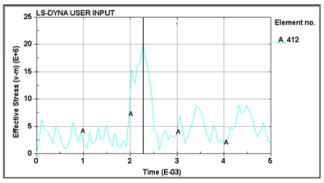
For the research of rotating collision process of driven-structure deformation, it is mainly consider that the position and the value of maximum stress. While finite element model of collision moment is shown in fig.10, stress distribution at the impacting moment is also in fig.11.

Though the simulation result, it is obviously to show that maximum stress generated at the collision contact point (that is element 412). Calculated the time of impacting process between plate and selector lever and obtained the collision moment is 0.0022s. Finally, Stress distribution in the process of impacting contact point is shown in fig.11, and maximum stress is 20.5e6Pa.

4.3 Compared with experiment result

While the flow velocity is 5.9m/s, strain original time history at the experiment collision point and stress time history of simulation collision point are obtained as shown in table 4. Maximum stress of sample collision point from calculating is 15.48e6Pa, and maximum stress from simulating is 20.5e6Pa. Resultant moments of experiment plate and simulation plate are respectively 0.21 $N \cdot m$ and 0.4 $N \cdot m$, it is clearly to acquire that the simulation results are close to experiment.

Table 4. Comparison of experimental and simulation results

History of strain vs. rotation angle at collision point (experiment)	History of strain vs. rotation angle at collision point (simulation)
	
Maximum strain value (experiment)	Maximum strain value (simulation)
15.48 Mpa	20.5 Mpa
Maximum torque applied on plate(experiment)	Maximum torque applied on plate (simulation)
0.21 $N \cdot m$	0.4 $N \cdot m$

5. Conclusions

In this paper, a new experiment platform developed by our lab for simulation and experiment research on fluid-structure coupling problems is introduced. This device could be used to measure the dynamics response of the moving underwater structure under impact loading, such as the transient rotating angle, impact strain, inlet fluid speed, and so on. Compared the simulation results from LS-DYNA and FlUENT with the measurement results, a simulation algorithm suitable for engineering application is provide, and some key coupling control parameters were investigated.

References

[1] Bukac, Martina; Canic, Suncica. Longitudinal displacement in viscoelastic arteries:A novel fluid-structure interaction computional model, and experimental validation. Mathematical biosciences and engineering, 2013, 10(2): p. 295-318.

[2] Tian,Fang-Bao,Dai, Hu,Luo,Haoxiang. Fluid-structure interaction involving large deformations: 3D simulations and applications to biological systems. Computational physics, 2014, 258: p. 451-469.

[3] Xing J T,Xiong Y P,Tan M.Developments of a mixed finite element substructure-subdomain method for fluid-structure interaction dynamics with applications in maritime engineering. Engineering for the maritime environment, 2009, 223(M3): p. 399-418.

[4] Falkovich, Gregory. Fluid Mechanics (A short course for physicists).Cambridge University Press, 2011.

[5] Kundu, Pijush K,Cohen, Ira M. Fluid Mechanics (4th revised ed.).Academic Press, 2008.

[6] Yan Ke,Ge Pei-qi,Bi Wen-bo.Vibration characteristics of fluid-structure interaction of conical spiral tube bundle.Hydrodynamic, 2010, 22(1): p. 121-128.

[7] Liu Demin,Liu Xiaobing.Vibration analysis of turbine based on fluid-structure coupling.Chinese journal of mechanical engineering, 2008, 21(4): p. 40-43.

[8] Han Li. Fluid-Structure Interaction in Fluid-driven deformable Structure. Beijing:Beihang university, 2014.