Fluid-Structure Interaction (FSI) of a Three-Bladed Marine Propeller

Onur Alpay[b], Özgür Demir[a], Yavuz Hakan Özdemir[a], Seyfettin BAYRAKTAR[a], Tamer YILMAZ[a]

[a]*Department of Naval Architecture and Marine Engineering, Yildiz Technical University*
[b]*Department of Mechanical Engineering, Yildiz Technical University*
34349 Besiktas, Istanbul, TURKEY

Abstract In this paper, the pressure loads on a three-bladed marine propeller is investigated numerically by fluid-structure interaction (FSI). Computational fluid dynamics (CFD) analysis is performed to obtain pressure field around the propeller using commercially available package of FLUENT 6.3.26 which uses finite volume method (FVM) as discretization scheme. Calculation of the stresses is done by using MSC NASTRAN that uses finite element method (FEM). Besides these a special pre-post processor code of MSC SOFY was used during mesh transformation process. In SOFY, a closed volume from the seed CFD mesh prepared and filled with tetrahedral solid elements. It is observed that main stress gradients were seen between the connecting tips of the blade and main carrying hub. The value of the stress was calculated as 12.2 MPa. This value is quite far from the stress limit of the propeller

Keywords: CFD, FEM, fluid-structure interaction (FSI), marine propeller, stress analysis, pressure field

1. Introduction

It is well known that an interaction occurs when a fluid interacts with a solid structure. It is general fluid mechanics rule that exerting pressure on a structure may cause deformation in the structure. Such interaction has been attracted more consideration recently because it is a crucial in the design and analysis procedures in marine engineering systems. It allows propellers to function, sails on marine vehicles to provide thrust, etc. Fluid and solid dynamical systems have been solved independently, however, for problems where there is sufficient coupling between the two systems, such separation is not possible and the resultant systems are invariably too complex to solve analytically. CFD is essential in predicting the behavior of such systems.

To determine the effects of fluid-structure interaction for a given system, engineering design often involves extensive experimental testing. However, it should be noted here that, experiments are costly and time consuming. Due to improvements in the numerical models, techniques and developments in the computing power with low prices, numerical simulation has become more established in the design process to support or even replace experimental testing. It requires consideration all mathematical models to solve a FSI problem numerically. For numerical simulation discretization methods such as FEM or the FVM should be applied. Depending on the application area, this procedure is commonly referred to as CFD or computational structure dynamics (CSD). The computation of the FSI involves both disciplines, CFD and CSD [1]. There are two methods to analyze the FSI problems, finite element method (FEM) and boundary element method (BEM). A wide list of references can be found in the bibliography [2 that contains the list of references that had been published in the scientific journals, conferences, etc., from 1995 to 1998. Dynamic deformation of structures under underwater impulsive loading was presented by Espinosa et al. [3]. Stella et al. [4] reported a numerical study of a hot flow through a pipe with flexible structures inside. All numerical simulation was performed by using CFD code of FLUENT while ADINA, a commercial code was used for the structural analyses. A procedure to transfer the interaction effects between the fluid and structure components was reported. Among many few studies on FSI to solve marine engineering problems, the paper of Zhang and Suzuki [5] should be refereed. They analyzed the liquid cargo filled tank during ship collision using the

ISSN-1223-7221 © 2008 Ovidius University Press

arbitrary Lagrangian-Eularian (ALE) finite element method. As aforementioned, FSI has been used commonly in most engineering disciplines. The effect of water hammer in the thick-walled pipes was investigated using FSI by Tijsseling [6] for this purpose a one-dimensional mathematical model was developed which based on conventional water-hammer and beam theories. Storm loading and pus-over analyses were introduced by using a code which was developed and reported by Bienen and Cassidiy [7].

Structural effects of varying water pressure are investigated in this study. To achieve this task, a CFD code, Fluent, was used to calculate the pressure field on the blades. CDF code uses the Finite Volume Method (FVM) rather than Finite Element Method (FEM). FVM depends on the Euler theory of kinematics. In this theory a stable volume of fluid is our focus of consideration. However in classic Lagrange Kinematics Theory we are interested in each single mass and motion characteristics of it. Calculation of pressure in a constant volume approach brings a great difference between FEM and FVM models. In a FVM model the fluid volume is the point of interest. Therefore there are no volume elements inside of the propeller. Volume elements are only modeling the surface of the propeller. Since it is interested in stress field on the propeller as a FEM result, it should be modeled to the inside of the propeller with tetrahedral solid elements. Although this technique is easier for using the same surface mesh and transferring the pressure fields on nodes, it does not have the capability to model the hub volume. Due to this limitation stress at the blade-hub connecting were neglected. With tetrahedral solid elements, stresses could be calculated.

2. Computational

In this paper, a three-bladed marine propeller with diameter of D=0.8 m was investigated numerically. The pressure and the stresses field distribution on the propeller were calculated using FVM and FEM, respectively. In Figure 1 the propeller under investigation is shown.

CFD Solver

CFD simulation was performed with a commercially available package of FLUENT 6.3.26. As turbulence closure standard k-ε turbulence (SKE) model was used with standard wall functions. The fluid around the propeller was assumed as incompressible, steady, and three-dimensional.

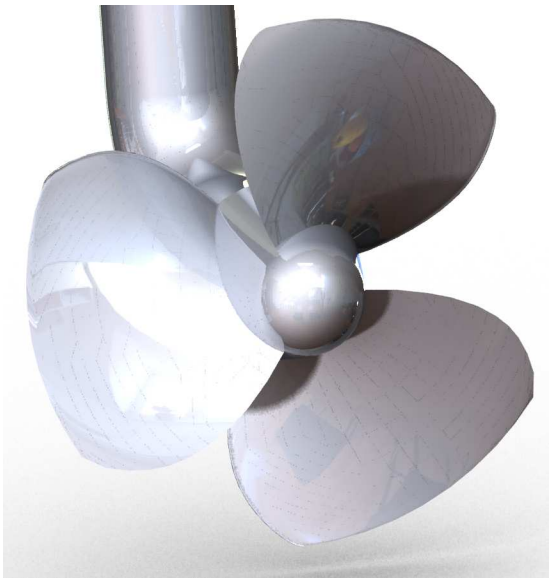


Fig.1: Propeller under investigated.

As it is well known that SKE is a semi-empirical model based on model transport equations for the turbulence kinetic energy, k (Eq.1) and its dissipation, ε (Eq. 2)

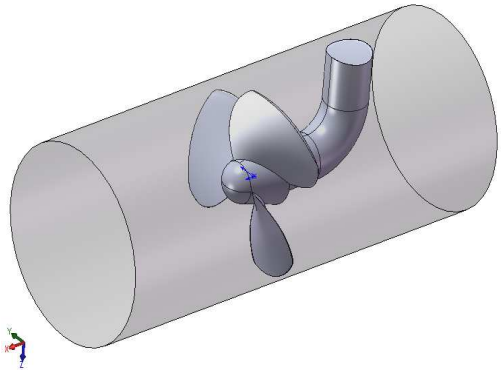
$$\frac{\partial \rho k}{\partial t}+\frac{\partial (\rho u_i k)}{\partial x_i}=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\mathrm{Pr}_k}\right)\frac{\partial k}{\partial x_i}\right]+ \quad (1)$$
$$\mu_T G-\rho\varepsilon+S_{k,p}$$

$$\frac{\partial \rho \varepsilon}{\partial t}+\frac{\partial (\rho u_i \varepsilon)}{\partial x_i}=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_T}{\mathrm{Pr}_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right]+ \quad (2)$$
$$\frac{\varepsilon}{k}\left(C_1\mu_T G-C_2\rho\varepsilon\right)+S_{\varepsilon,p}$$

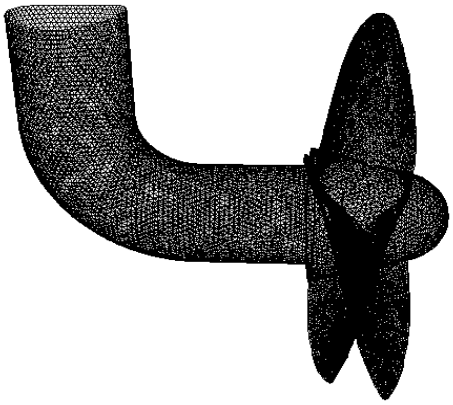
where C_1 and C_2 are additional dimensionless model constants; Pr_k and Pr_e are the turbulent Prandtl numbers for kinetic energy and dissipation, respectively; $S_{k,p}$ and $S_{e,p}$ are source terms for the kinetic energy and turbulent dissipation; and the turbulent production rate (G) is defined in Eq. 3:

$$G = \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} - \frac{1}{\rho^2} \frac{\partial \rho}{\partial x_j} \frac{\partial \rho}{\partial x_J} - \frac{2}{3} \left(\frac{\rho k}{\mu_T} + \frac{\partial u_i}{\partial x_J} \right) \frac{\partial u_J}{\partial x_j} \quad (3)$$

Computational cylindrical domain and unstructured mesh applied to the propeller is showed in Figure 2.



(a)



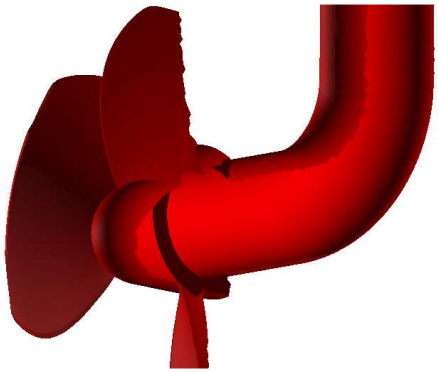
(b)

Fig.2: Computational domain (a) and mesh structure applied to the propeller for CFD analysis (b).

Total length of the cylinder is three times of the propeller diameter. All the grid structure was generated by using GAMBIT 2.4.6 a pre-processor of FLUENT. Totally 3,500,000 hexahedral mesh elements are used for the whole domain including propeller. Boundary conditions were set to simulate flow around the rotating propeller in open water. Pressure inlet and pressure outlet was specified as inlet and outlet conditions. Absolute pressure value was taken as 0 (zero) for both inlet and outlet.

CSD Solver

MSC.NASTRAN, a commercially available FEM code, was used for calculation of the stress area. There are two main steps for preparing the FEM model for NASTRAN. In first step, tetrahedral solid mesh prepared based on the surface mesh of CDF code.



(a)

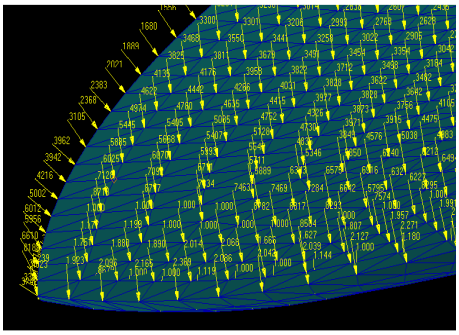


(b)

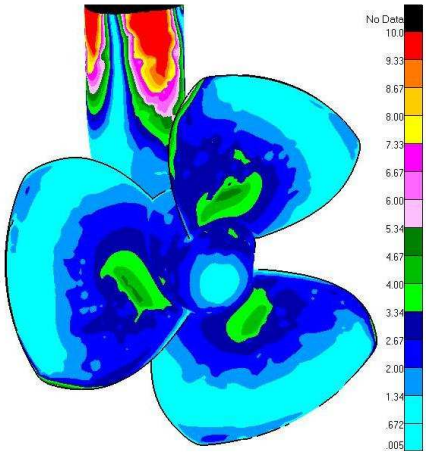
Fig.3: Surfaces of the propeller for VFM analysis (a) and solid mesh calculated with FEM (b).

The challenge here was to prepare the solid mesh exactly in the same size and same orientation of the surface mesh seed to precisely transfer the pressure field calculated in FLUENT. To do this, a special pre-post processor code, MSC.SOFY was used. In SOFY, a closed volume from the seed CDF mesh prepared and filled with tetrahedral solid elements as

seen in Figure 3. The second step was to import the pressure field from Fluent and apply correct pressure value on each node. In another pre-post processor, MSC.PATRAN, pressure field on surface nodes was applied on nodes of solid mesh. Pressure equivalent forces and applied force contours on the propeller is shown in Figure 4.



(a)



(b)

Fig.4: Pressure equivalent forces varying on propeller blades (a) and applied force contours on entire propeller (b).

In many engineering applications FEM is an important tool. Accuracy, low cost and applicability of this method are the main reasons of this importance. Strength and dynamic calculations of structural systems is a large application area of FEM. Another practical side of the FEM is the displacement approach. In classical strength of materials calculation of the support reactions is a common challenge. Thus statically indefinite problems are much harder to solve. On the other hand, in FEM applications, over definition lowers the degree of freedom of the system and makes the problem mush easier to solve. Below, formulation of a tetrahedral element which used in this study is given. The element and global displacement vectors are,

$$q = [q_1, q_2, q_3 ..... q_n]^T \quad (4)$$

$$Q = [Q_1, Q_2, Q_3 ..... Q_n]^T \quad (5)$$

where N is the total number of degrees of freedom for the structure, three per node. Four Lagrange-type shape functions are defined; N_1, N_2, N_3, N_4 where shape function N_i has a value of 1 at node i and is zero at the other three nodes. Shape function is given as follows (Eq.6):

$$\begin{aligned} N_1 &= \xi \\ N_2 &= \eta \\ N_3 &= \zeta \\ N_4 &= 1 - \xi - \eta - \zeta \end{aligned} \quad (6)$$

The displacements u,v,w at x can be written in terms if the unknown N nodal values as given in Eq.7 :

$$u = N q \quad (7)$$

The element strain energy in the total potential is given in Eq. 8:

$$U_e = \frac{1}{2}\int_e \varepsilon^T D\varepsilon \, dV = \frac{1}{2} q^T k^e q \quad (8)$$

where the element stiffness matrix k^e is given by Eq. 9:

$$k^e = V_e B^T D B \quad (9)$$

The potential term associated with body force is given in Eq. 10.

$$\int_e uTf dV = qT \iiint N^T f \det J \; d\xi \, d\eta \, d\zeta = q^T f^e \quad (10)$$

Using formula in Eq. 10, Eq. 11 can be obtained.

$$f^e = \frac{V_e}{4}[\,fx, fy, fz, fx, fy, fz, \ldots.. fz]^T \qquad (11)$$

The element traction load vector T^e is given by Eq. 12:

$$T^e = \frac{A_e}{3}[Tx, Ty, Tz, Tx, Ty, Tz, Tx, Ty, Tz, 0, 0, 0] \quad (12)$$

The energy and Galerkin approaches yield the set of equations

$$KQ = F \qquad (13)$$

After the above equations (Eq. 4-13) are solved, the element nodal displacements q can be obtained. Since $\sigma = D\varepsilon$ and $\varepsilon = Bq$, the element stresses are given by

$$\sigma = DBq \qquad (14)$$

In this study FEM was used for calculation of the pressure induced stress contours on the blades of the propeller.

3. Results and Discussions

Figure 5 shows the displacement contours of propeller. According to deformation fringe, largest displacements were observed at the tip of the most loaded blade. The 0.179 mm displacement is the magnitude of this displacement.

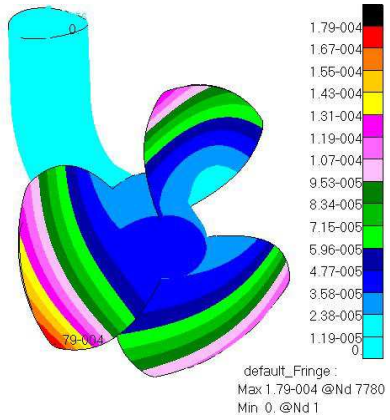
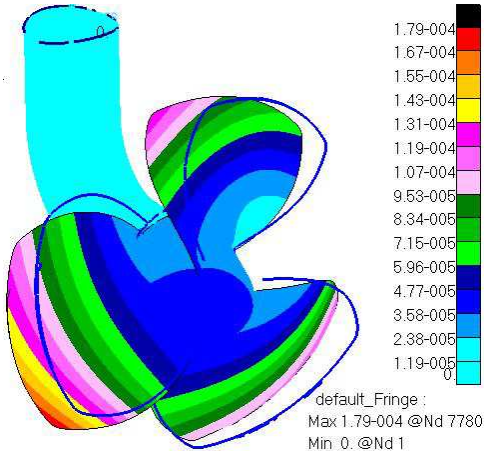
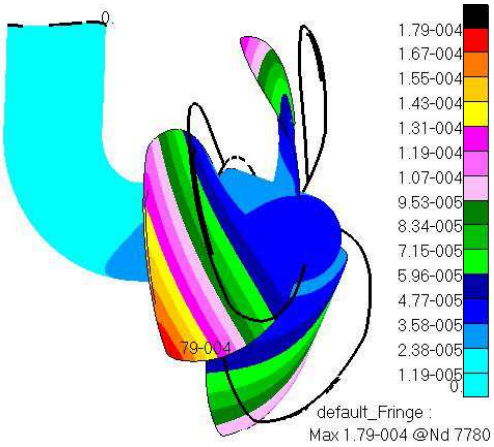


Fig.5: Displacement contours.

Following plots in Figure 6 are well describing the deformed shape of the propeller. Blades are bending opposite direction of the fluid flow.



(a)



(b)

Fig.6: Deformed shape of the propeller, a) front view, b) side view.

As can be seen from Figure 7, main stress gradients are between the connecting tips of the blade and main carrying hub. The magnitude is about 12.2 MPa. This value is quite far from the stress limit of the propeller. As seen here the failure of the propeller is not depend on the standard working loads. Front side of the connecting tip of the blades is under tension stress

while the back side is under compression stress as seen in the Figure 7.

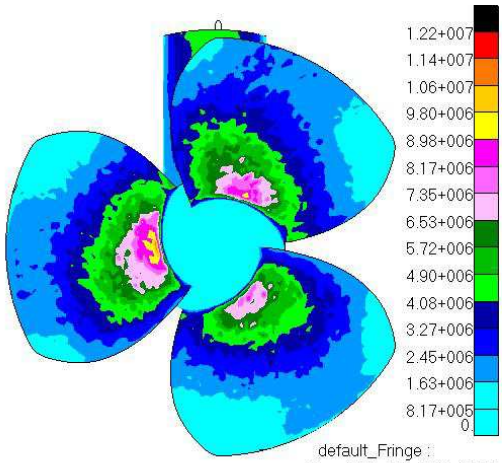


Fig.7: Stress gradients on the front side of the propeller.

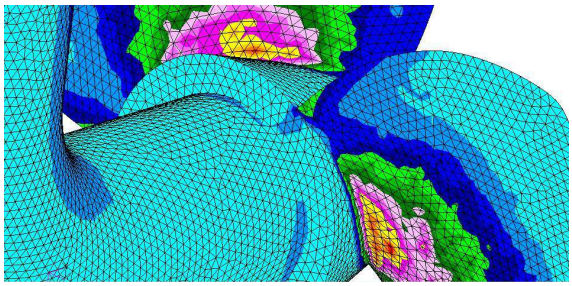


Fig.8: Stress gradients on the back side of the propeller.

4. Conclusions

A numerical procedure for the structural analysis of a three-dimensional nature has been developed to provide a reliable solution to the problem of determining the strength of a three-bladed marine propeller. A highly skewed propeller blade under pressure distributions was analyzed and stress results were presented. Pressure loads on the marine propeller was calculated with a commercial CFD code of FLUENT which uses FVM. The flow around the propeller was modeled by using standard k-ε turbulence model. Totally 3500000 mesh elements were used for the whole domain.

Obtained pressure loads on the propeller were then used for determining the stresses on the propeller by using MSC NASTRAN and MSC PATRAN. Mesh transformation was achieved by MSC SOFY. It is found that main stress gradient was between the connecting tips of the blade and main carrying hub.

6. References

[1] Michler, C., "Efficient Numerical Methods for Fluid-Structure Interaction", Report 2005.

[2] Mackerle,J.,"Fluid-Structure Interaction Problems, Finite Element and Boundary Element Approaches", Finite Element in Analysis and Design 31, 231-240, 1999.

[3]Espinosa, H.D., Lee, S., Moldovan, N., "A Novel Fluid Structure Interaction Experiment to Investigate Deformation of Structural Elements Subjected to Impulsive Loading", Experimental Mechanics 46, 805-824, 2006.

[4] Stella, F., Giangi, M., Paglia, F., Casata, A., Simone, D., "A Numerical Simulation of Fluid-Strcutre Interaction in Internal Flows", Numerical Heat Transfer, Part B, 47, 403-418, 2005.

[5] Zhang, A., Suzuki, K., "Numerical Simulaiton of Fluid-Structure Interaction of Liquid Cargo Filled Tank during Ship Collision Using the ALE Finite Element Method" International Journal of Crashworthiness, 11:4, 291-298, 2005.

[6] Tijsseling, A.S., "Water hammer with fluid structure interaction in thick-walled pipes", Computers &Fluids 85, 844-851, 2007.

[7] Bienen, B., Cassidiy, M.J., "Advances in the Three-Dimensional Fluid-Structure-Soil Interaction Analysis of Offshore Jack-up Structures", Marine Structures 19, 110-140, 2006.