FLUID STRUCTURE INTERACTION ANALYSIS ON ELLIPSOIDAL SUBMERGED BODIES BY USING CFD

Y. Amar Babu*, Dr. S. Adinarayana**, Md. Yousuf Ali***.

* Department of Mechanical Engg. M.V.G.R. College of Engineering

Abstract- The interaction between a semi-rigid structure and the surrounding fluid environment is a sober issue in the stability investigation of different foil availabilities and design of aircraft blade machinery. To study about these types of phenomena, it requires the modeling of both fluid and structural domains. The two methods available to calculate the Fluid–Structure Interaction (FSI) effects in the time domain are the strongly coupled or partially coupled governing equations. In the fully coupled model, the flow field and structure respond simultaneously by exchanging the aerodynamic forces and structural displacement. Logically, the fully coupled model is rigorous in the physical sense, because the structural displacement responds instantly to the forces imposed by the fluid. In this algorithm, the fluid and structural analyses equations are combined together to form a unified set of equations. Then it is solved and integrated simultaneously in time domain to update all the variables including those at the fluid–structure interface. This implies solving the complete system in one step; hence there is no information transfer in fully-coupled method. The fully coupled algorithm usually requires an almost complete rewrite of the CFD and CSD codes into one single coupled code. In this project, the modeling is done in Creo Parametric 2.0 and Computational fluid dynamics analysis and fluid structure interaction (FSI) is done in ANSYS 15.0 solver. The static structural analysis is carried out initially. The flow analysis of various angles of attack are carried out and FSI is done for the respective models. The later and the former are compared for pressure distributions and velocity magnitudes for various angles of attack. The pressure coefficients and velocity magnitudes are comparably good for Fluid structure Interaction.

Index Terms- Creo parametric 2.0, Fluid Structure Interaction, Pressure Distributions, Velocity Magnitudes.

I. INTRODUCTION

Fluid–structure interaction (FSI) is the interaction of some movable or deformable structure with an internal or surrounding fluid flow. Fluid–structure interactions can be stable or oscillatory. In oscillatory interactions, the strain induced in the solid structure causes it to move such that the source of strain is reduced, and the structure returns to its former state only for the process to repeat. Fluid–structure interactions are a crucial consideration in the design of many engineering systems, e.g. aircraft, engines and bridges. Failing to consider the effects of oscillatory interactions can be catastrophic, especially in structures comprising materials susceptible to fatigue. Tacoma Narrows Bridge (1940), the first Tacoma Narrows Bridge, is probably one of the most infamous examples of large-scale failure. Aircraft wings and turbine blades can break due to FSI oscillations. Fluid–structure interaction has to be taken into account for the analysis of aneurysms in large arteries and artificial heart valves.

A reed actually produces sound because the system of equations governing its dynamics has oscillatory solutions. The dynamic of reed valves used in two stroke engines and compressors is governed by FSI. The act of "blowing a raspberry" is another such example. Fluid–structure interactions also occur in moving containers, where liquid oscillations due to the container motion impose substantial magnitudes of forces and moments to the container structure that affect the stability of the transport system in a highly adverse manner. The propagation of a pressure wave through an incompressible fluid in a flexible tube is as shown in the Fig. 1

Fig. 1 Propagation of a Pressure wave
II. PROBLEM IDENTIFICATION

The present study made computational prediction of Fluid Structure Interaction of submerged bodies using commercially available RANS code, Fluent, with suitable User Defined Functions (UDFs) as well as a CFD Code, developed in-house. The fluid and structure has been created with appropriate dimension for understanding the interaction between the fluid and the structure. Fluid structure interaction is carried out at different angle of attack. The differences in fluid forces acting on a structural member for both the ways of coupling have been analyzed. The pressure coefficient and the velocity magnitude are compared with interaction and without interaction.

III. MODELING OF ELLIPSOIDAL MODEL

The modeling of the Ellipsoidal Body is done in Creo Parametric 2.0.

Introduction to Creo Parametric:

Creo Parametric is a computer graphics system for modeling various mechanical designs and for performing related design and manufacturing operations. The system uses a 3D solid modeling system as the core, and applies the feature-based, parametric modeling method. In short, Creo Parametric is a feature-based, parametric solid modeling system with many extended design and manufacturing applications.

Creo Parametric is the first commercial CAD system entirely based upon the feature-based design and parametric modeling philosophy. Today many software producers have recognized the advantage of this approach and started to shift their product onto this platform.

Creo Parametric was designed to begin where the design engineer begins with features and design criteria. Creo Parametric's cascading menus flow in an intuitive manner, providing logical choices and pre-selecting most common options, in addition to short menu descriptions and full on-line help. This makes it simple to learn and utilize even for the most casual user. Expert users employ Creo Parametric's "map keys" to combine frequently used commands along with customized menus to exponentially increase their speed in use. Because Creo Parametric provides the ability to sketch directly on the solid model, feature placement is simple and accurate.

The model is as shown in the figure 1 as shown below:

Fig 1. Ellipsoidal Model
The drawing Specifications taken are as shown in the Figure 2 below:

![Fig. 2 Drawing Specifications for the Ellipsoidal Model.](image)

IV. ANALYSIS OF ELLIPSOIDAL MODEL

The analysis of the Ellipsoidal Model is done in Ansys 15.0 and the analysis reports are as shown below. The geometry and the mesh model in Ansys are as shown in the Fig.3 and Fig. 4 below respectively.

![Fig. 3 Geometry of the Ellipsoidal Model](image)  ![Fig.4 Mesh of the Ellipsoidal Model](image)

Analysis of Ellipsoidal Model:

The deformation and Equivalent Stress reports for the Ellipsoidal Models are are as shown in the Fig. 5 and Fig. 6 respectively.
The Pressure Coefficient and Velocity magnitude reports of Zero degree for the Ellipsoidal Models are as shown in the Fig. 7 and Fig. 8 respectively.

Also the analysis is carried out for the fluid structure interaction treating as a one way FSI. The Pressure Coefficient and Velocity magnitude reports of Zero degree with structural interaction for the Ellipsoidal Models are shown in the Fig. 9 and Fig. 10 respectively.
V. RESULTS AND DISCUSSION

The Pressure Coefficients for Various Angles of attack is as shown in the Table

<table>
<thead>
<tr>
<th>Angle of Attack</th>
<th>Fluid Flow Analysis (Max) (m/s)</th>
<th>Fluid Structure Interaction (Max) (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0.8231</td>
<td>1.6702</td>
</tr>
<tr>
<td>2</td>
<td>0.8232</td>
<td>1.8243</td>
</tr>
<tr>
<td>4</td>
<td>0.8233</td>
<td>2.0036</td>
</tr>
<tr>
<td>6</td>
<td>0.8252</td>
<td>2.7496</td>
</tr>
<tr>
<td>8</td>
<td>1.1195</td>
<td>3.7181</td>
</tr>
</tbody>
</table>

The Velocity Magnitudes for Various Angles of attack is as shown in the Table

<table>
<thead>
<tr>
<th>Angle of Attack</th>
<th>Fluid Flow Analysis (Max) (m/s)</th>
<th>Fluid Structure Interaction (Max) (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>5.1093</td>
<td>5.1676</td>
</tr>
<tr>
<td>2</td>
<td>5.4836</td>
<td>5.2018</td>
</tr>
<tr>
<td>4</td>
<td>5.9352</td>
<td>5.3801</td>
</tr>
<tr>
<td>6</td>
<td>6.4099</td>
<td>5.5350</td>
</tr>
<tr>
<td>8</td>
<td>6.8643</td>
<td>5.6772</td>
</tr>
</tbody>
</table>

In the fluid structure interaction analysis of different angles of attack we observed the values of pressure coefficients are better than fluid flow analysis pressure coefficient values. The fluid structure interaction analysis is comparatively good. The work is performed by method of coupling ie. One way coupling. This is implemented by creating a large domain in which the structural member is placed in an appropriate position. The meshing of the ellipsoidal body is obtained good mesh quality which is best in the flow analysis.

And the same way we observed the velocity magnitude for the flow analysis and the fluid structure interaction analysis. The fluid structure interaction has much more max values at different angles of degree. The values of pressure coefficient and velocity magnitude are tabulated in the above. By comparing these fluid structure interaction analysis and the flow analysis results we notified that the fluid structure interaction analysis is best way to study the fluid flow over the structure. There are many instances of structural damages of such platforms due to extreme weather conditions like hurricanes. These circumstances have a created an awareness in offshore industries on things like structural durability and reliability and the requirements of setting a safer air gap.

By this analysis we conclude that the fluid structure interaction analysis is applicable for study more advancement in the underwater submerged bodies for new development in marine engineering.
VI. CONCLUSION

As a part of real time application model, an ellipsoid body of 1:10 is taken into consideration for evaluation. The model is run at a free attack at a velocity of 5m/s initially and pressure distribution along the run is observed. The fluid flow analysis for different angle of attacks is done. Pressure and velocity distributions are observed. The structure analysis is carried out. The fluid structure interaction for the model is run and pressure and velocity distributions are observed after CFD simulation considering it as an FSI application. The pressure coefficients and velocity magnitudes are comparably good for Fluid structure Interaction. As the angle of attack increases, the pressure coefficient is linearly increasing.

**Scope for future work:** 1. As CFD results were found in good agreement with standard or experimental results, more complex shapes of interest can be developed in order to find the variations of pressure coefficients and velocity distributions. 2. The present work contributes a one way method to the problem solution. Different other types of methodologies can be used to solve the problem using CFD Simulation.

**REFERENCES**


**AUTHORS**


**Second Author** – Dr. S. Adinarayana, Professor, HOD, Department of Mechanical Engg. M.V.G.R.College of Engineering.


**Correspondence Author** – Y. Amar Babu, yamarbabu@gmail.com