



NRC Publications Archive Archives des publications du CNRC

A numerical study of fluid structure interaction of a flexible submerged cylinder mounted on an experimental rig

Cakir, Erkan; Akinturk, Ayhan; Allievi, Alejandro

This publication could be one of several versions: author's original, accepted manuscript or the publisher's version. / La version de cette publication peut être l'une des suivantes : la version prépublication de l'auteur, la version acceptée du manuscrit ou la version de l'éditeur.

For the publisher's version, please access the DOI link below. / Pour consulter la version de l'éditeur, utilisez le lien DOI ci-dessous.

Publisher's version / Version de l'éditeur:

<https://doi.org/10.1115/OMAE2015-42219>

Proceedings of the ASME 2015 34th International Conference on Ocean, Offshore and Arctic Engineering, OMAE2015, May 31-June 5, 2015, St. John's, Newfoundland, Canada, 2015-05-31

NRC Publications Record / Notice d'Archives des publications de CNRC:

<https://nrc-publications.canada.ca/eng/view/object/?id=99ced429-7a07-427e-acf8-be02ced03ed0>

<https://publications-cnrc.canada.ca/fra/voir/objet/?id=99ced429-7a07-427e-acf8-be02ced03ed0>

Access and use of this website and the material on it are subject to the Terms and Conditions set forth at

<https://nrc-publications.canada.ca/eng/copyright>

READ THESE TERMS AND CONDITIONS CAREFULLY BEFORE USING THIS WEBSITE.

L'accès à ce site Web et l'utilisation de son contenu sont assujettis aux conditions présentées dans le site

<https://publications-cnrc.canada.ca/fra/droits>

LISEZ CES CONDITIONS ATTENTIVEMENT AVANT D'UTILISER CE SITE WEB.

Questions? Contact the NRC Publications Archive team at

PublicationsArchive-ArchivesPublications@nrc-cnrc.gc.ca. If you wish to email the authors directly, please see the first page of the publication for their contact information.

Vous avez des questions? Nous pouvons vous aider. Pour communiquer directement avec un auteur, consultez la première page de la revue dans laquelle son article a été publié afin de trouver ses coordonnées. Si vous n'arrivez pas à les repérer, communiquez avec nous à PublicationsArchive-ArchivesPublications@nrc-cnrc.gc.ca.



OMAE-42219

A NUMERICAL STUDY OF FLUID STRUCTURE INTERACTION OF A FLEXIBLE SUBMERGED CYLINDER MOUNTED ON AN EXPERIMENTAL RIG

Erkan Cakir*

Department of Ocean and
Naval Architectural Engineering
Memorial University of Newfoundland
St. John's, A1B 3X5
erkan.cakir@mun.ca

Ayhan Akinturk

National Research Council, OCRE
St. John's, A1B 3T5
ayhan.akinturk@nrc-cnrc.gc.ca

Alejandro Allievi

Memorial University of Newfoundland
St. John's, A1B 3X5
alejandro.allievi@gmail.com

ABSTRACT

The aim of the study is to investigate VIV effects, not only on a test cylinder but also on the experimental rig being towed under water at a prescribed depth and operating speeds. For this purpose, a numerical Multi-Physics model was created using one way coupled analysis simultaneously between the Mechanical and Fluent solvers of ANSYS software package. A system coupling was developed in order to communicate force data alternately between the solvers with the help of automatic mapping algorithms within millesimal time periods of a second. Numerical investigation into the dynamic characteristics of pressure and velocity fields for turbulent viscous fluid flow along with structural responses of the system, stressed the significance of time and space scales for convergence and accuracy of our Finite Volume (FV) CFD calculations.

INTRODUCTION

It is a known fact that, with the development of the oil and gas industry to deeper waters and harsher environments, engineering problems require more complex solutions with predictable performances under the aforementioned conditions. Due to the increasing complexity of today's engineering solutions the need for accuracy of numerical simulations has become more significant. This concern necessitates the consideration of more than one physical factor [1]. Moreover, with the increasing

significance of these factors' influence on the response of the system, fluid-structure interactions carry greater importance. In consideration of modeling and computational calculations, FSI problems can be the most challenging simulations. Due to the complexity of the multi-physics problems it is generally challenging to solve them analytically. Thus, numerical simulations or experiments are required to deal with these problems. In addition, the choice of an appropriate solution method for an FSI problem is crucial while developing a numerical model. Monolithic approach aims to solve all flow and displacement equations simultaneously. However, partitioned approach uses the more specified and advanced solvers which were developed only for structural or fluid simulations. FSI problems investigate the structural and hydrodynamics effects along with their close relations within the system. Due to shape dependency of hydrodynamic forces of structures and the dependency of structural deformations as a result of hydrodynamic loads, current problems require exclusive analysis for each application. Considering the wide use of cylindrical shaped structures such as risers, moored and tethered structures along with sub sea umbilicals and spar hulls, our study focuses on FSI investigation of the assembly composed of a fixed flexible cylinder with different material properties mounted on a test rig designed for future experimental investigations. We have used a two way coupled partitioned approach by the aid of ANSYS Workbench environment using Mechanical and Fluent solvers. Since one way coupled partitioned approach is based on the coupling of

*Address all correspondence to this author.

fluid and structural solvers in order to send only force values within the solvers, this approach lacks displacement effects on the hydrodynamic loads. It can only be used when the deformations of the system are small or can be neglected. On the other hand, two way coupled method sends deformation values to the fluid solvers by upgrading the mesh of fluid domain for every desired time step. This method provides a more realistic and advanced simulation of the physical problem especially when the deformations are large and their effects on the hydrodynamic loads are highly significant.

Experimental Rig

Designing an experiment for investigating a fluid-structure interaction requires high precision and full system integrity. The study was focused on a flexible cylinder mounted on an experimental rig. Thus, it is crucial to have a reliable system for measuring the flow forces around the cylinder with minimum vibrations in order to avoid unintentional instabilities on the test subject. Given that, the structural endurance of the system opposing to the fluid forces is essential as well. The flow around the cylinder should be isolated from reflecting flows and any kinds of disturbances. For this purpose, we aimed to design an experimental rig that will tow our test subject, a circular cylinder with 0.0015 m diameter and 0.60 m length, in different depths and speeds. The cylinder was designed to be attached to a plate that is 0.24 m long and 0.90 m wide in order to simulate the ocean floor in different operating conditions. The plate was designed to be attached to the carriage with a main support consisting of a 0.1016 m diameter pipe and a fin that was used in order to delay vortex formations around the main support. However, towing a submerged large plate without being exposed to severe vibrations under the speed of 4 m/s is a challenge resulting in the requirement of a sturdy supporting system. Considering this, we have placed two cylindrical front supports to the system in order to provide more stability without disturbing the flow around the cylinder. Finally, the plate was strengthened with a frame stiffener system from the bottom of the plate. Taking these challenges into account, the design demonstrated in Figure 1 was developed after several optimization trials.

NUMERICAL MODEL

Structural Model

A finite element model was created in order to investigate structural responses of the experimental rig and the test subject cylinder. Considering the difficulties of FEA on hyper elastic materials due to the large deformations and their ability to withstand greater strains than conventional materials [2], default structural steel model from ANSYS Material Database [3] was used as material for the plate, support system and rigid cylinder. For the

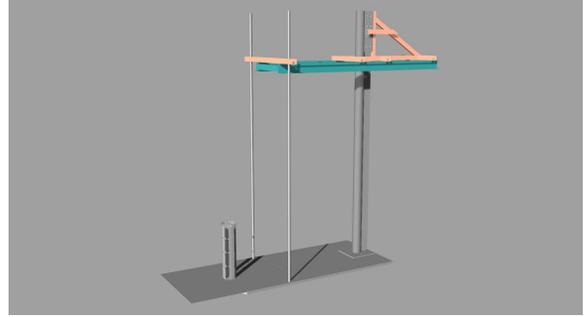


FIGURE 1. DESIGNED THE EXPERIMENTAL RIG

flexible cylinder, structural steel model was modified by decreasing its Young's modulus providing a more stable flexible material. For the purpose of simplifying the structural module, front supports were introduced as fixed support condition onto the region of their attachment point with the plate. Due to the software limitations and non-linear stresses the model was created using solid elements. For this purpose, SOLID 186 and SOLID 187 type elements were used in ANSYS Mechanical. These elements have 20 and 10 nodes respectively and both elements support large deflections and large strain capabilities along with plasticity, hyper elasticity, creep and stress stiffening [4].

Calculating the coupled fluid structure analysis includes the solution of the structural problem for each time step simultaneously. Thus, using the best solver with the most efficient resources was critical. Moreover, considering the need for stability and endurance of the experimental rig under designated loading conditions, a mesh convergence study was conducted based on the maximum von-Mises stress criteria in order to develop the most accurate and computationally affordable numerical finite element model.

Figure 2 shows the final mesh of the structural model for the experimental rig including the plate, cylinder, cylinder cap and the support system that contains 61375 elements with 265000 nodes.

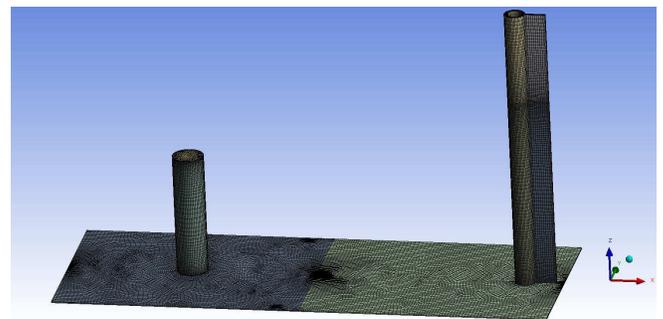


FIGURE 2. FLUID DOMAIN SIDE VIEW

One of the challenges of FSI problems is their demand for computational resources. In order to obtain numerical results in the desired time, solver performances of the both Mechanical module and Fluent in ANSYS was critical. Even though the CFD portion of the FSI problem requires more computational resources, due to the size of the structural solution and the necessity for repetition at the end of each coupling iteration, the performance of the Mechanical solver was also crucial. A numerical benchmark test for comparison of solver type vs. calculation time was conducted to choose the most robust solver for this particular problem. Even though direct solver was more robust for static structural analysis of the experimental rig's responses, due to the extensive output file sizes and non-linearities in the system, Pre-conditioned Conjugate Gradient (PCG), an iterative solver, was used due to efficiency and memory and storage usage. It was also evident that for this specific solver the use of eight parallel processors gave the most speed-up. Some of the governing stress equations that mechanical solvers address can be shown as follows [5]:

$$\{\sigma\} = [D]\{\varepsilon\} \quad (1)$$

in the equation, [D] represents elastic stiffness matrix and $\{\sigma\}$ represents stress vector which can also be written as:

$$\{\sigma\} = [\sigma_x \sigma_y \sigma_z \sigma_{xy} \sigma_{yz} \sigma_{xz}] \quad (2)$$

$\{\varepsilon\}$ stands for total strain vector which can also be written as:

$$\{\varepsilon\} = \text{total strain vector} = [\varepsilon_x \varepsilon_y \varepsilon_z \varepsilon_{xy} \varepsilon_{yz} \varepsilon_{xz}] \quad (3)$$

ε_{xy} , ε_{yz} , ε_{xz} represent the shear strains. Shear strains can be expressed as follows by the aid of Shear modulus and stresses:

$$\varepsilon_{xy} = \frac{\sigma_{xy}}{G_{xy}} \quad (4)$$

$$\varepsilon_{yz} = \frac{\sigma_{yz}}{G_{yz}} \quad (5)$$

$$\varepsilon_{xz} = \frac{\sigma_{xz}}{G_{xz}} \quad (6)$$

whereas shear modulus can be written as:

$$G = \frac{E}{2(1 + \nu)} \quad (7)$$

Finally von-Mises Stresses can be found by the following formula:

$$\sigma_e = \left[\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}{2} \right] \quad (8)$$

Fluid Flow Model

A single phase fluid flow model was developed and Realizable k-epsilon model was used to solve turbulent flow. Dimensions of the fluid domain used for this study are shown in Figure 3 and 4.

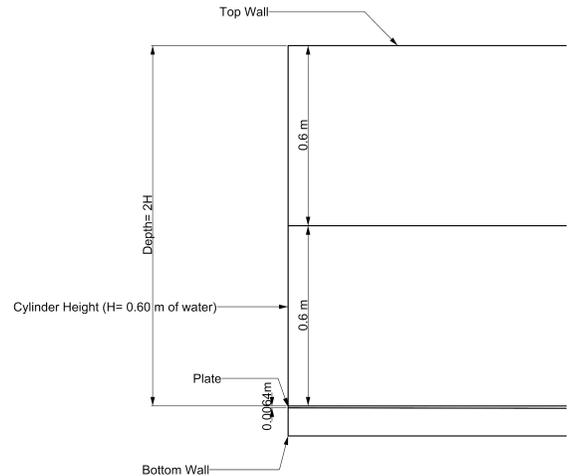


FIGURE 3. FLUID DOMAIN SIDE VIEW

A velocity-inlet and pressure-outlet type boundary condition was used for inlet and outlet of the fluid domain. The distance between the inlet boundary and cylinder was 1.5 m where it is 4 m to the outlet. Flow specifications were defined as velocity (4 m/s) and direction. Furthermore, default values for turbulence intensity and viscosity ratio were used. No slip wall boundary conditions were used for bottom, top and side walls of the domain. Figure 5 shows the CFD mesh of the fluid domain in different perspectives along with the mesh around the cylinder. CFD mesh consists of 2 million elements with an average of 0.04 skewness and 0.98 orthogonal quality.

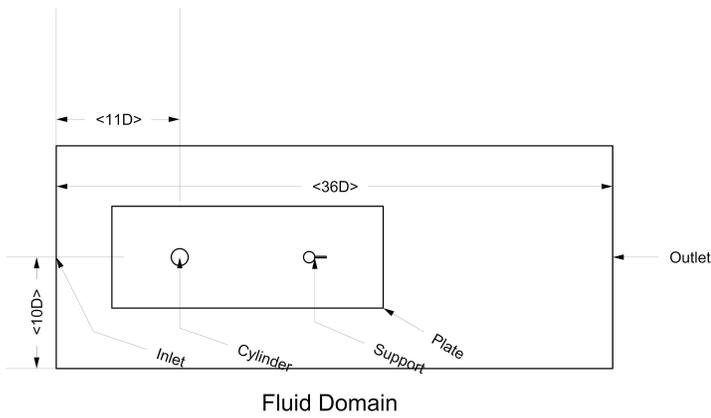


FIGURE 4. FLUID DOMAIN TOP VIEW

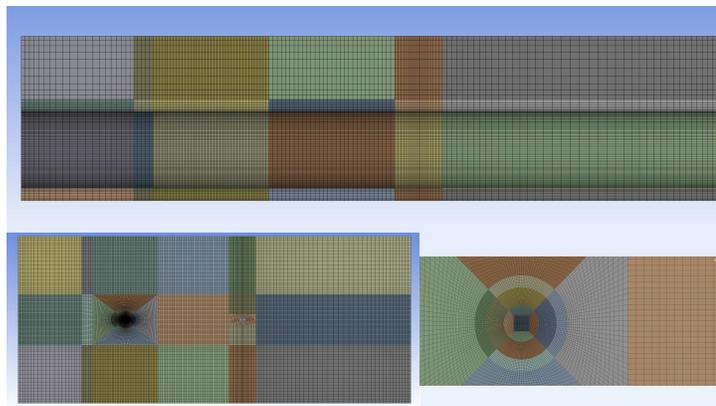


FIGURE 5. CFD MESH

Turbulence Model

Turbulence is a phenomenon causing disturbances in the flow as a relation of space and time due to the large velocity gradients. It is known that there are still unresolved complications especially with the high Reynolds number and Mach numbers. Reynolds Averaged Navier-Stokes Equations can be shown as below [6]:

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\frac{\mu}{\rho} \frac{\partial u_i}{\partial x_j} \right) \quad (9)$$

Besides, today's computational resources are inadequate to solve the turbulence for engineering problems by using Direct Numerical Simulations (DNS) due to excessive computational costs. Consequently, averaging methods such as Reynolds averaging have to be employed. On the other hand, there is no conventional turbulence model that can be used for all flow scenarios while simulating a turbulent flow. Thus, the preference of turbulence model depends on the factors of available time, computational power, physics of the flow and the desired accuracy level. There are several turbulence models for numerical modelling such as, zero equation (Prantls Mixing Length, Cebeci-Smith Model), one-equation model (Spart-Allmaras), two-equation models ($k-\epsilon$, $k-\omega$), second order closure (Reynolds stress) and algebraic stress models. Reynolds Averaged Navier-Stokes (RANS) models are computationally the most economical ones that are also widely used for simulating industrial flows due to their ability to provide acceptable accuracy. These models solve the problem by the help of two additional transport equations and Eddy-Viscosity to estimate Reynolds Stresses. In this study, $k-\epsilon$ model with standard wall functions is used due to its robustness and economy. Moreover, the mesh around the cylinder was refined in order to provide a wall y^+ value between 30 and 100.

Coupling Fluid and Structural Solvers

Transferring data between solvers happens in every time step if the solutions are converged or they reach to the maximum number of iterations. By considering the physics of our problem a time step of 0.001 second was used. FSI regions were chosen separately for cylinder, plate and support and data transfers was created for each of the plate, support and cylinder surface regions in order to send force data. Between the FSI regions of Mechanical and CFD, %100 mapping were established within ANSYS System Coupling. For 1 Way coupled analysis, Fluent solves the equations for whole fluid domain and sends the results on the cylinder, plate and support. This concludes a time step of calculation in case of convergence of the both solutions.

NUMERICAL RESULTS

The results of one way coupled FSI analysis with a single phase turbulent flow around the experimental rig were presented. Lift and drag coefficients around the test object, a circular cylinder, were plotted on Figure 6.

For the structural investigation of the rig's response, deformations were plotted in a bigger scale of 5.5/1. Figure 7 and 8 present the deformations on the entire rig. It can be clearly seen that at the initial condition of the flow, hydrostatic forces cause 0.006 m deformation on the plate.

For the CFD perspective of problem, the local values of the velocity and pressure of the flow are shown by the aid of horizontal and vertical planes. Results are presented for 0 to 3 seconds of

flow time in Figures 9-16. Figure 9 shows the results from initial time to 0.55 seconds where the first vortex shedding occurs around the cylinder. Figure 10 demonstrates the results between 0.75 to 1 seconds. Figure 11 and 12 present the velocity magnitude patterns of 1.65- 2 seconds and 2.5 to 3.0 seconds respectively. The velocity magnitude plots that were created on the horizontal and vertical planes demonstrate that the vortex shedding for this case starts at 0.55 seconds. Furthermore, pressure plots were shown in Figure 13 represent the values between 0 to 0.55 seconds. Figure 14, 15 and 16 demonstrates the pressure plots at 0.75-1 s, 1-1.65 and 2.5-3 s respectively. One way Coupled FSI results are shown in the Figures 17-20. Velocity of the flow and the resulting hydrodynamic loads along with their effects on the structural model were shown as von-Mises stresses. These results were investigated at 0 s, 0.5 s, 1 s and 1.5 s of the solution time.

CONCLUDING REMARKS

A turbulent flow regime past a circular cylinder attached to an experimental rig was investigated. The focus of the study is to understand structural responses of the experimental rig and flexible cylinder under a prescribed depth and speed along with the effects of flexibility of the cylinder on hydrodynamic loading conditions. Results have shown that due to high drag and oscillating lift forces and their effects on the rig, the simplifications within structural numerical model have to be reconsidered. In the process of developing the structural FE model stiffener frame was not included. However, due to the exceeded predicted deformations which can cause vibrations on the test subject and decrease the reliability of the experiments, the stiffener system should also be taken into account. Moreover, in pursuance of more accurate results for simulating the towing tank flow, open channel flow with VOF method should be used. Finally, the prediction of the experimental rig’s performance and the test subject’s structural behaviour under hydrodynamic loads can be investigated with a two way coupled analysis more accurately and realistically. For this purpose, re-meshing and smoothing dynamic mesh methods can be used.

REFERENCES

[1] Hans-Joachim Bungartz, M. S., ed. *Fluid-Structure Interaction Modelling, Simulation, Optimisation*. p. V.
 [2] Xiao-Yan, R. M. On Stress Analysis for a Hyperelastic Material. Tech. rep., Sulzer Carbomedics Inc.
 [3] ANSYS, 2013. *Engineering Data User Guide*, release 15 ed. Canonsburg. See also URL <http://www.ansys.com>.
 [4] ANSYS, 2009. *Element Reference Guide*, release 12.1 ed. Canonsburg. See also URL <http://www.ansys.com>.
 [5] ANSYS, 2013. *Mechanical APDL Theory Guide*, release 15 ed. Canonsburg. See also URL <http://www.ansys.com>.

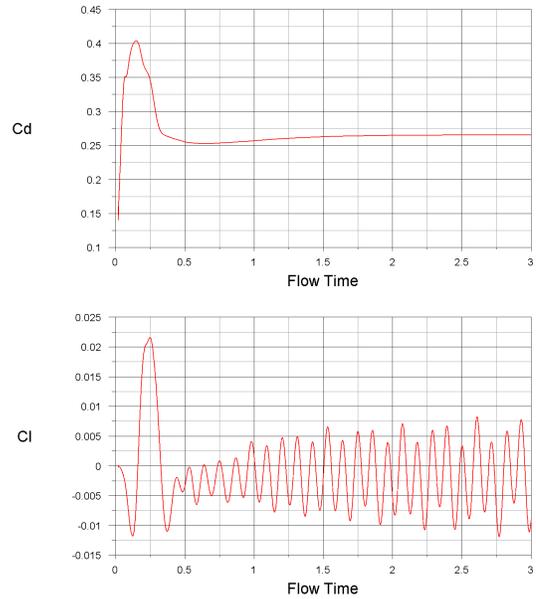


FIGURE 6. LIFT AND DRAG COEFFICIENTS FOR CYLINDER (RE=600,000)

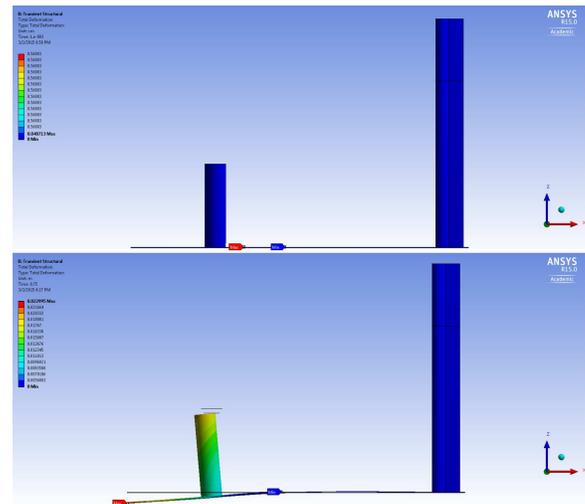


FIGURE 7. DEFORMATIONS ON THE EXPERIMENTAL RIG (0 s - 0.75 s)

[6] Chung, T. J., ed. *Computational Fluid Dynamics*. ch. 21, p. 679.

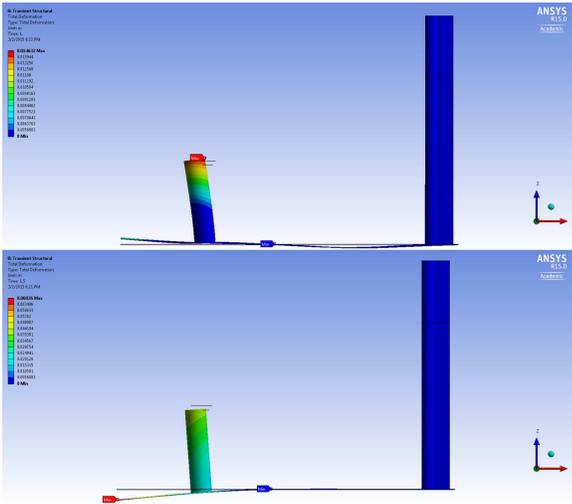


FIGURE 8. DEFORMATIONS ON THE EXPERIMENTAL RIG (1 s - 1.5 s)

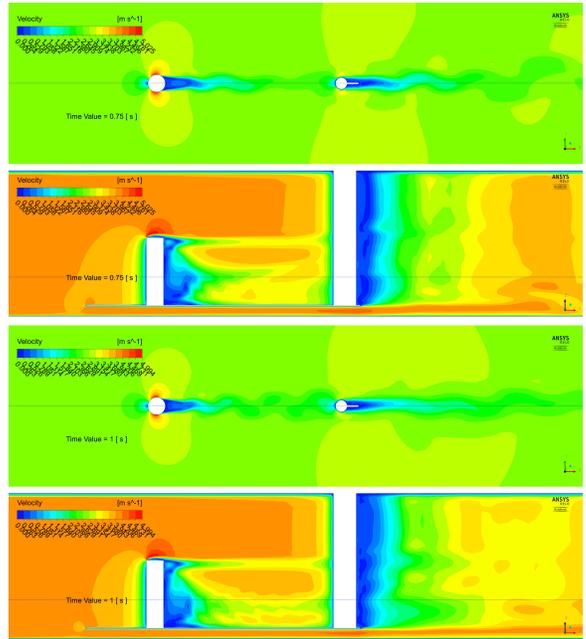


FIGURE 10. VELOCITY PROFILE OF THE EXPERIMENTAL RIG (0.75-1.0 SECONDS)

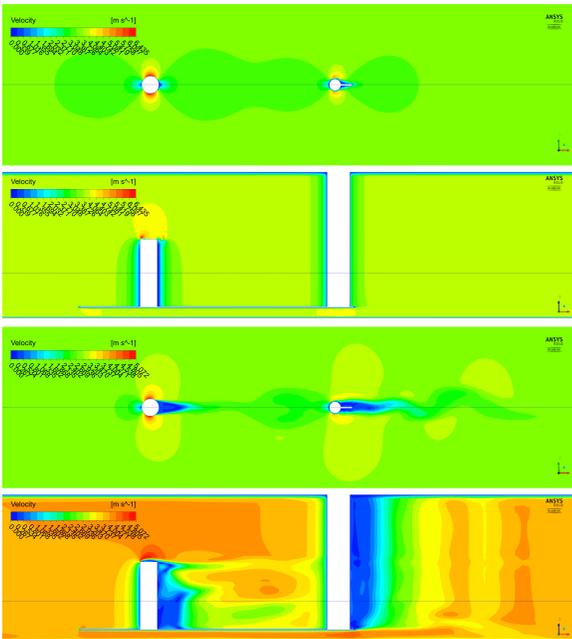


FIGURE 9. VELOCITY PROFILE OF THE EXPERIMENTAL RIG (0-0.55 SECONDS)

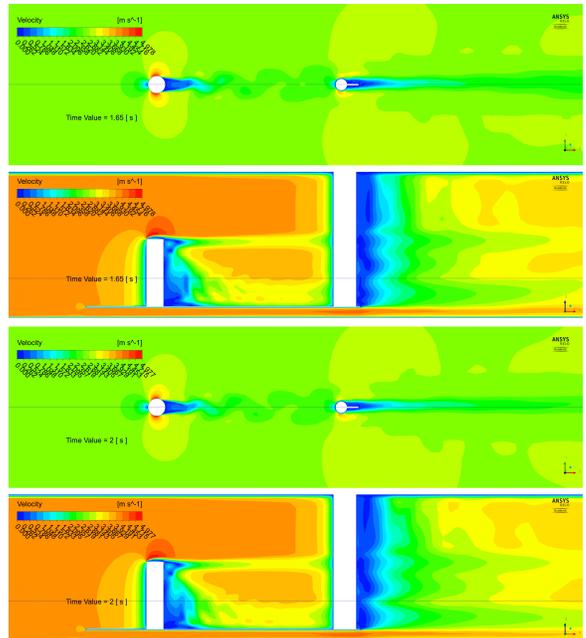


FIGURE 11. VELOCITY PROFILE OF THE EXPERIMENTAL RIG (1.0-1.65 SECONDS)

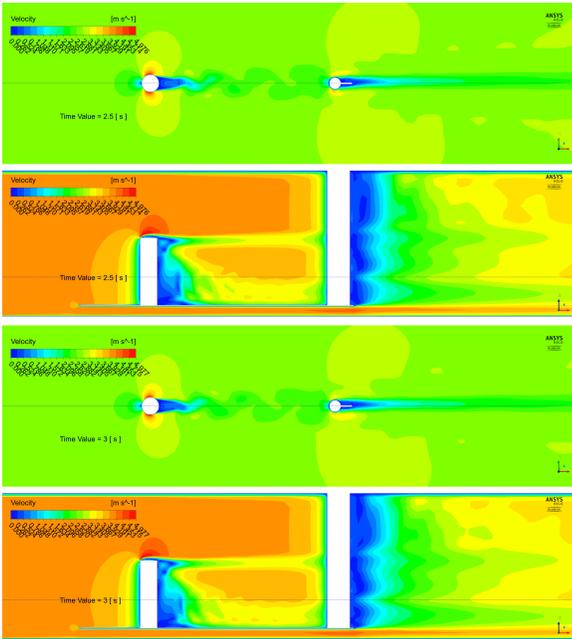


FIGURE 12. VELOCITY PROFILE OF THE EXPERIMENTAL RIG (2.5-3.0 SECONDS)

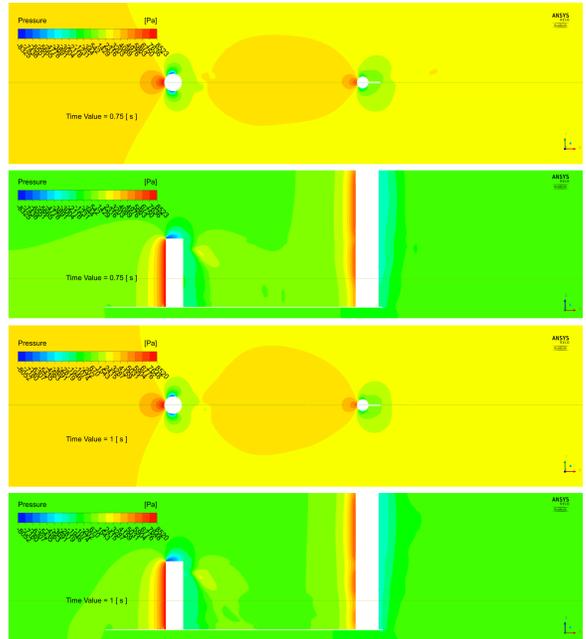


FIGURE 14. PRESSURE PROFILE OF THE EXPERIMENTAL RIG (0.75-1.0 SECONDS)

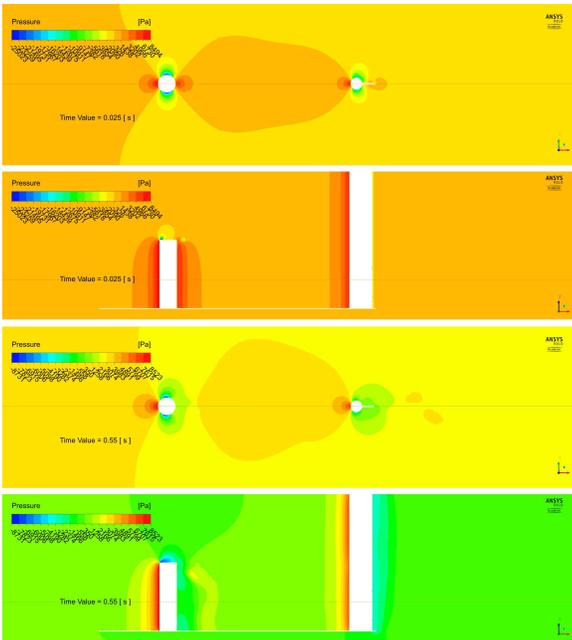


FIGURE 13. PRESSURE PROFILE OF THE EXPERIMENTAL RIG (0-0.55 SECONDS)

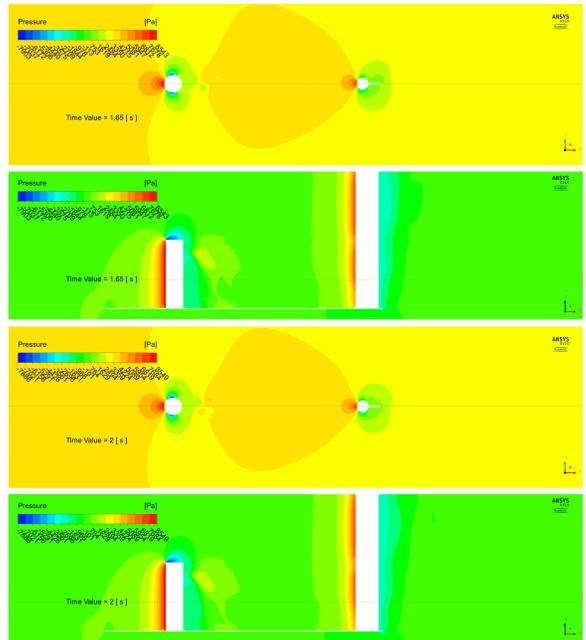


FIGURE 15. PRESSURE PROFILE OF THE EXPERIMENTAL RIG (1.0-1.65 SECONDS)

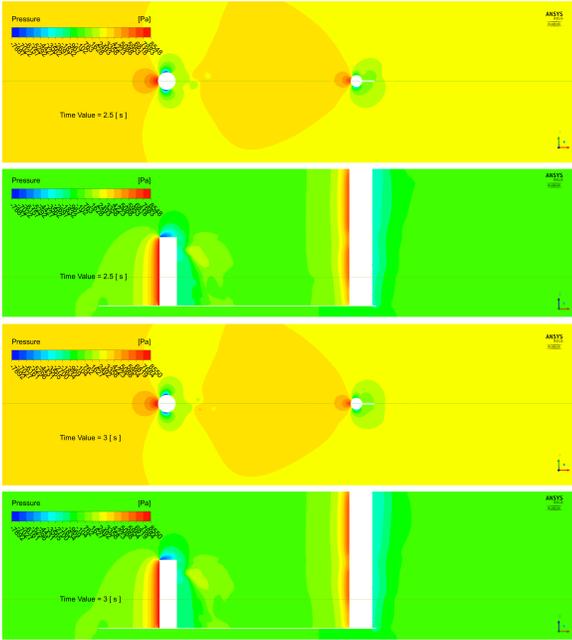


FIGURE 16. PRESSURE PROFILE OF THE EXPERIMENTAL RIG (2.5-3.0 SECONDS)

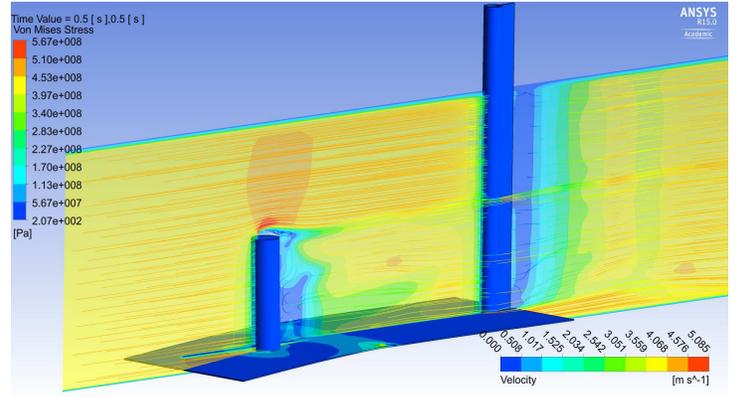


FIGURE 18. 1WAY FSI T=0.5 SEC

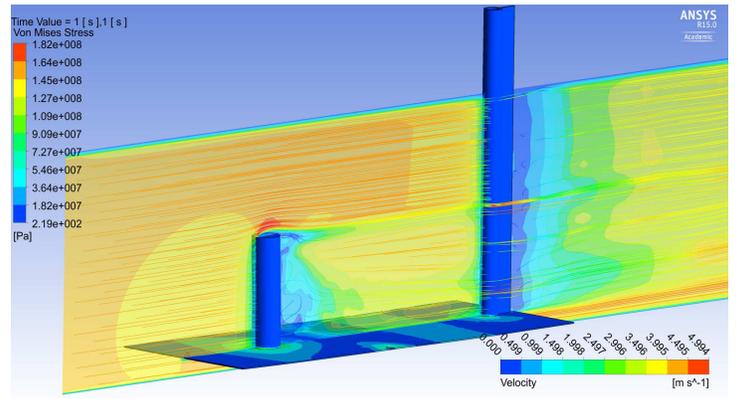


FIGURE 19. 1WAY FSI T=1 SEC

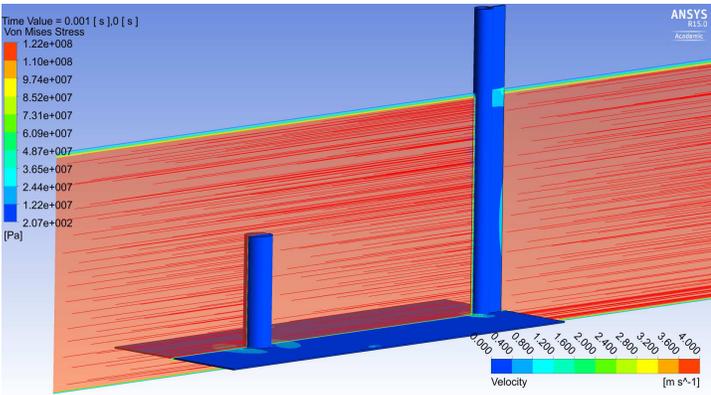


FIGURE 17. 1WAY FSI T=0 SEC

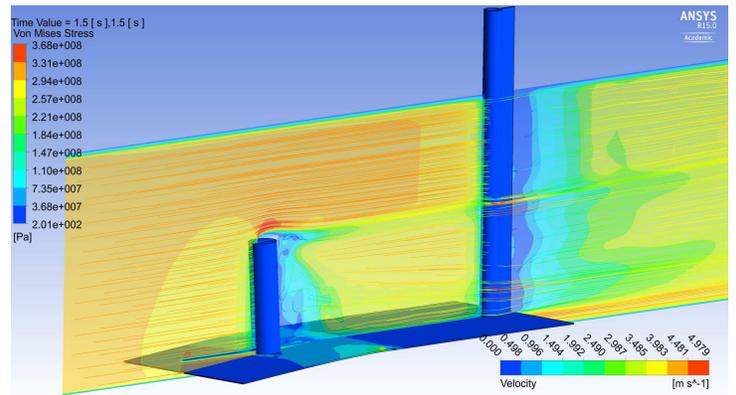


FIGURE 20. 1WAY FSI T=1.5 SEC