Model of the hydrodynamic loads applied on a rotating half-bridge belonging to a circular settling tank

A E Dascalescu¹, G Lazaroiu¹, A A Scupi² and E Oanta²
¹ Politechnica’ University of Bucharest, Faculty of Power Engineering, 313 Splaiul Independentei, Sector 6, 060042, Bucharest, Romania
² Constanta Maritime University, Faculty of Naval Electro-Mechanics, 104 Mircea cel Batran Street, 900663, Constanta, Romania
E-mail: eoanta@yahoo.com

Abstract. The rotating half-bridge of a settling tank is employed to sweep the sludge from the wastewater and to vacuum and send it to the central collector. It has a complex geometry but the main beam may be considered a slender bar loaded by the following category of forces: concentrated forces produced by the weight of the scrapping system of blades, suction pipes, local sludge collecting chamber, plus the sludge in the horizontal sludge transporting pipes; forces produced by the access bridge; buoyant forces produced by the floating barrels according to Archimedes’ principle; distributed forces produced by the weight of the main bridge; hydrodynamic forces. In order to evaluate the hydrodynamic loads we have conceived a numerical model based on the finite volume method, using the ANSYS-Fluent software. To model the flow we used the equations of Reynolds Averaged Navier-Stokes (RANS) for liquids together with Volume of Fluid model (VOF) for multiphase flows. For turbulent model k-epsilon we used the equation for turbulent kinetic energy k and dissipation epsilon. These results will be used to increase the accuracy of the loads’ sub-model in the theoretical models, i.e. the finite element model and the analytical model.

1. Introduction
The settling tank is a part of a wastewater treatment plant used to gravitationally settle the sludge, to vacuum and to send it to the sludge tank. According to figure 1, the rotating half-bridge sweeps the sludge using the system of scrapping blades – 3, suction pipes – 4 and 5, local sludge collecting chamber – 6 and the sludge transporting horizontal pipes – 7. All the forces produced by these parts together with the forces produced by the access bridge and of main bridge the are weights. Along the vertical direction there are also oriented the buoyant forces produced by the floating barrels – 6.

2. Theoretical background
The motion of the main bridge in the wastewater represents a fluid-structure interaction problem. To solve this problem we used the following theoretical background that also represents the state of the art of this kind of problems.

Attempts to model the fluid flow started 150 years ago, initially by implementing the ideal flow using the Eulerian calculus model. The most rigorous set of equations which model the monophasic flow of a fluid is the Navier-Stokes equations, which in a vector form may be expressed as:
\[ F - \frac{1}{\rho} \nabla p + \eta \frac{\Delta \vec{v} + \nu}{3} \nabla(\nabla \vec{v}) = \frac{d\vec{v}}{dt}. \]  

(1)

**Figure 1.** Main components of the rotating bridge.

In a scalar form we have

\[ F_x = \frac{1}{\rho} \frac{\partial p}{\partial x} + \nu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) + \frac{\nu}{3} \frac{\partial \theta}{\partial t} = \frac{\partial v}{\partial t} + \frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} + \frac{\partial v_z}{\partial z}, \]  

(2)

\[ F_y = \frac{1}{\rho} \frac{\partial p}{\partial y} + \nu \left( \frac{\partial^2 v_y}{\partial x^2} + \frac{\partial^2 v_y}{\partial y^2} + \frac{\partial^2 v_y}{\partial z^2} \right) + \frac{\nu}{3} \frac{\partial \theta}{\partial t} = \frac{\partial v_y}{\partial t} + \frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} + \frac{\partial v_z}{\partial z}, \]  

(3)

\[ F_z = \frac{1}{\rho} \frac{\partial p}{\partial z} + \nu \left( \frac{\partial^2 v_z}{\partial x^2} + \frac{\partial^2 v_z}{\partial y^2} + \frac{\partial^2 v_z}{\partial z^2} \right) + \frac{\nu}{3} \frac{\partial \theta}{\partial t} = \frac{\partial v_z}{\partial t} + \frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} + \frac{\partial v_z}{\partial z}. \]  

(4)

where \( F \) is the unitary mass force, \( \vec{v} \) is the fluid’s velocity, \( p \) is the pressure, \( \rho \) is the density, \( \eta \) is the dynamic viscosity and \( \theta \) is the velocity’s divergence. Until now, there were not conceived analytic solutions of these equations, [1].

Usually, the Navier-Stokes equations are solved together with the continuity equation:

\[ \frac{d\rho}{dt} + \rho \nabla \vec{v} = 0. \]  

(5)

In our model we don’t use in an explicit way the continuity equation, the solver taking implicitly into account this equation. Moreover, in our model the Navier-Stokes equations are used in the context of a multi-phase flow, being applied in an individual way, for each of the components: water, sludge and air. To accomplish this, we used the volume of fluid (VOF) method, as it is implemented in the ANSYS Academic software application. The volume of fluid is based on the aspect that the fluids are not interpenetrating. Each new phase added to the model is introducing a new variable regarding the volume fraction of the new phase in the computational cell, the summation of all volume fractions being equal to unity. In this way, in every volume cell the variables and properties are either characteristic to one of the phases, or specific to a mixture of the phases, with respect to the volume fraction values [2]. For every \( q \) phase, the volume fraction equation has the following form:

\[ \frac{1}{\rho_q} \left[ \frac{\partial}{\partial t} \left( \alpha_q \rho_q \right) + \nabla \cdot \left( \alpha_q \rho_q \vec{v} \right) \right] = S_{\alpha_q} + \sum_{p=1}^{n} \left( \dot{m}_{qp} - \dot{m}_{pq} \right) \]  

(6)

where: \( \dot{m}_{qp} \) is the mass transfer from phase \( q \) to phase \( p \) and \( \dot{m}_{pq} \) is the mass transfer from phase \( p \) to phase \( q \); \( \alpha_q \) is the volume fraction of the phase \( q \) and \( S_{\alpha_q} \) is a specific constant.
Depending on the phase, the Reynold number is in the range $\text{Re} \in (344; 7064)$, therefore a turbulent model is appropriate. The $k - \varepsilon$ model together with the VOF model were successfully used in several models, [3][4], the results being accurate.

The $k - \varepsilon$ model is a semi-empirical model based on the model transport equation for the turbulence kinetic energy ($k$) and its dissipation rate ($\varepsilon$). The realizable sub-model of the $k - \varepsilon$ model usually uses: the equation for turbulent kinetic energy (7), dissipation epsilon (8) and the energy equation (9) [1]:

\[
\frac{\partial}{\partial t} (\rho_m \cdot k) + \nabla \cdot (\rho_m \cdot \tilde{v}_m \cdot k) = \nabla \cdot \left( \frac{\mu_{t,m}}{\sigma_k} \nabla k \right) + G_{k,m} - \rho_m \cdot \varepsilon, \tag{7}
\]

\[
\frac{\partial}{\partial t} (\rho_m \cdot \varepsilon) + \nabla \cdot (\rho_m \cdot \tilde{v}_m \cdot \varepsilon) = \nabla \cdot \left( \frac{\mu_{t,m}}{\sigma_{\varepsilon}} \nabla \varepsilon \right) + \frac{\varepsilon}{k_c} \cdot \left( C_{k} \cdot G_{k,m} - C_{2\varepsilon} \cdot \rho \cdot \varepsilon \right), \tag{8}
\]

\[
\frac{\partial}{\partial t} (\rho \cdot E) + \nabla \cdot (\tilde{v} \cdot (\rho \cdot E + p)) = \nabla \cdot \left( k_{eff} \cdot \nabla T - \sum_j h_j \cdot J_j + \frac{\varepsilon}{\tau_{eff} \cdot \tilde{v}} \right) + S_h. \tag{9}
\]

The mean density is

\[
\rho_m = \sum_{i=1}^{N} (\alpha_i \cdot \rho_i), \tag{10}
\]

The mean velocity is

\[
\tilde{v}_m = \frac{\sum_{i=1}^{N} (\alpha_i \cdot \rho_i \cdot \tilde{v}_i)}{\sum_{i=1}^{N} (\alpha_i \cdot \rho_i)}. \tag{11}
\]

The turbulent (or eddy) viscosity is

\[
\mu_{t,m} = \rho_m \cdot C_\mu \cdot \frac{k^2}{\varepsilon}, \tag{12}
\]

where $C_\mu$ is a constant.

The generation of turbulence kinetic energy due to the mean velocity gradients is

\[
G_{k,m} = \mu_{t,m} \left( \nabla \tilde{v}_m \cdot (\nabla \tilde{v}_m)^T \right) : \nabla \tilde{v}_m, \tag{13}
\]

where $k_{eff}$ is the effective conductivity; $J_j$ is the fluid diffusion flux; $S_h$ is the heat due to the chemical reaction. In the equation (9) we have:

\[
E = h - \frac{p}{\rho} + \frac{\varepsilon^2}{2}, \tag{14}
\]

$h$ is the enthalpy; for ideal fluids (15) and for real fluid (16)

\[
h = \sum_j (Y_j \cdot h_j), \tag{15}
\]

\[
h = \sum_j (Y_j \cdot h_j) + \frac{p}{\rho}, \tag{16}
\]

\[
h = \int_{T_{ref}} c_{v,j} \cdot dT, \tag{17}
\]

where $T_{ref} = 298.15 \, K$.

In our model we disregard the energy equation because the temperature is not influencing the fluid-structure interaction phenomenon.
3. Discussion and results

The geometric model was developed in NX, figure 2, and then imported in ANSYS.

Figure 2. Geometric model developed in NX.

The detailed model was simplified because we were interested only in the fluid-structure interaction, i.e. in the submerged area of the rotating bridge. This simplified geometric model of the bridge was included in a calculus domain which is the settling tank, presented in figure 3.

Figure 3. Geometric model and the calculus domain in ANSYS.
The fluids’ domain has three phases: water, sludge and air. The water has a density of 1000 kg/m³ and represents 60% of the fluids’ domain. The sludge has a density of 2600 kg/m³ and it has 16% of the domain.

The domain was discretized in 5413929 computation cells with 1050026 nodes. The cells are small enough, in order to offer accurate results, the quality of the mesh being represented in figure 4. As it can be noticed the quality of most of the cells is over 75%.

The final solution presented in the following figure resulted after several attempts in which we used ICEM CFD and using various discretization methods in the implicit meshing application of ANSYS. Various meshing variants having a number of cells between 2 million and 5 million were created and tested.

The outer surfaces of the submerged bridge were selected and designated in a relevant way, in accord with their location and functional role. For each surface of the bridge, there was defined a corresponding surface of the fluids’ domain. This procedure is useful for a follow-up study regarding the structural effects of the hydrodynamic forces applied on the structure of the bridge.

The settings of the solver are: type – ‘pressure based’, velocity formulation – ‘absolute’ and time was set to ‘transient’. There was also set the gravitational acceleration useful to keep the phases in distinct locations. An ‘explicit’ model of ‘volume of fluid’ was chosen with ‘implicit body forces’ option selected. We also selected the ‘realizable’ k-epsilon model. The phases were defined in descending order with respect to their weight: sludge, water and air. The fluids were set to have a rotational motion around the centre of the tank with an angular velocity equal to the real velocity of the bridge – a revolution in 57 minutes.

At first we have separated the fluids taking into account their densities, the result being presented in the following figure.

Fluids were rotated using the previously mentioned angular velocity and after the solution converged, we checked the most important parameters of the phenomenon: gauge pressure, dynamic pressure and velocity magnitude. In the following figure it is presented the gauge pressure variation through a plane that intersects the central scrapping system as side view and top view of the domain.

![Figure 4. Mesh and quality of the mesh.](image-url)
In order to have some results that are closer to the real phenomenon we used second-order discretization schemes. Thus, we used second-order upwind finite volume scheme for the convective terms and second-order central finite volume scheme for the diffusion terms. We used the second-order discretization schemes because in other models they led to accurate results, in accord to the values resulted from the experimental studies, [5].

Results of the study are presented in the previous table. The values are to be compared with the results of an analytic approach which is in progress.

### Table 1. Results of the numerical model.

|                              | Surfaces [m²] | Forces [N] | Dynamic pressure [Pa] |
|------------------------------|---------------|------------|-----------------------|
| Inner scrapping system       | 5.6510796     | 644.216    | 113.99                |
| Central scrapping system     | 6.9427625     | 2163.08    | 311.55                |
| Outer scrapping system       | 8.2253095     | 2006.85    | 243.98                |

### 4. Conclusions

The paper investigates the hydrodynamic phenomena in a settling tank, being focused on the calculus of the pressure distribution onto the scrapping system of the half-bridge.
In general, a complex phenomenon may be studied using analytical, numerical and experimental studies. Because of the high complexity of the structure we couldn’t perform ‘in-situ’ relevant measurements. The first step was to model the fluid-structure interaction we used the ANSYS-Fluent software. We also have an ongoing study regarding the analytic model of the pressure distribution.

The results of this paper i.e. the pressure distributions represent the input data of a following study which will be dedicated to the creation of a more accurate model of the loads to be used in a structural study regarding the half-bridge.

Acknowledgement

Part of the investigation presented in the paper was done in the framework of the POSDRU 132397 project: Excellence in research by offering doctoral and postdoctoral scholarships – ExcelDOC in which Anca-Elena Dascalescu is involved.

The paper is based on the ideas developed in the MIEC2010 bilateral Ro-Md research project, E. Oanta, C. Panait, L. Lepadatu, R. Tamas, M. Constantinescu, I. Odagescu, I. Tamas, G. Batrinca, C. Nistor, V. Marina, G. Iliadi, V. Sontea, V. Marina, V. Balan, V. (2010-2012), “Mathematical Models for Inter-Domain Research with Applications in Engineering and Economy”, [7], under the guidance of the National Committee of Scientific Research, Romania, that is a follow-up set of studies of the ID1223 scientific research project: E. Oanta, C. Panait, B. Nicolescu, S. Dinu, A. Pescaru, A. Nita, G. Gavrila, (2007-2010), “Computer Aided Advanced Studies in Applied Elasticity from an Interdisciplinary Overview”, [6], under the guidance of the National University Research Council, Romania.

References

[1] http://www.claymath.org/millennium-problems/navier%E2%80%93stokes-equation, accessed: 10/02/2016.
[2] *** Ansys-Fluent v. 14.5 Theory guide, 2012.
[3] Sagau M, Panaitescu M, Panaitescu F V, Scupi A A 2013 A new considerations about floating storage and regasification unit for liquid natural gas, Procs of EEEAD, ISBN: 978-1-61804-211-8, pp 135-140.
[4] Panaitescu M, Dumitrescu G S, Scupi A A 2013 Sustainable pneumatic transport systems of cereals, Procs of EEEAD, ISBN: 978-1-61804-211-8, pp 128-134.
[5] Scupi A A, Avital E J , Dinu D, Williams J J R, Munjiza A Large eddy simulations of flows around a kite used as an auxiliary propulsion system, Journal of Fluids Engineering, Transactions of the ASME, 137 10, pp. 101301-1÷101301-8 doi: 10.1115/1.4030482.
[6] Oanta E, Panait C, Nicolescu B, Dinu S, Pescaru A, Nita A, Gavrila G 2007 Computed aided advanced studies in applied elasticity from an interdisciplinary perspective ID1223 CNCSIS Romania research project.
[7] Oanta E, Panait C, Lepadatu L, Tamas R, Batrinca G, Nistor C, Marina V, Iliadi G, Sontea V, Marina V, Balan V 2010 Mathematical models for inter-domain approaches with applications in engineering and economy, MIEC2010, ANCS Ro-Md scientific research project.