Studying and Analysis of Nonlinear Sloshing (Vibrating) of Interaction Fluid Structure in Storage Tanks

Ali I. Al-Zughaibi, Emad Q. Hussein, Farhan Lafta Rashid*

College of Engineering, University of Kerbala, Karbala, Iraq.

*Corresponding Author Email: ali.i@uokerbala.edu.iq; emad.dujaily@uokerbala.edu.iq; farhan.lefta@uokerbala.edu.iq

ABSTRACT

Sloshing (vibration) is an evolving phenomenon of fluid movement, with non-linear and highly random properties that affect the tank wall and can cause structural fatigue resulting in tank destruction. The main objective of the present work is to study slip behavior in rigid tanks that undergo sudden acceleration. ANSYS was used for three different tank filling depths as (30%, 50%, 70%) for 3D transient analysis and a multi-phase model was adapted to track the free surface of the liquid through the use of Volume of Fluid (VOF). A system coupling was modified for conjoining the fluid and solid domain. The dynamic meshing technique is used to simulate excitation sources via a User Defined Function (UDF) by developing a code in the C ++ environment to evaluate the interference.. This study showed the transient stress and deformation of the baffle increases by close to 55%, as a result of changing the partially filled liquid depths from 30% to 70%, whereas the magnitude of them was decreased about 10% when the tank using baffles, thus, it should be considering and has become an important outcome in tank design. The comparison of the simulation data with the previous measurement is performed to prove the effectiveness and reality of the Finite Element Methods (FEM) simulates model**.**

KEYWORDS

Sloshing, Liquid Tank, Baffles, Dynamic Meshing, Fluid Structure Interaction.

INTRODUCTION

Sloshing is a complex phenomenon with random and non-linear properties of fluid motion. There are several factors that affect the movement of palpitations, which can be summarized as follows: The frequency and amplitude of the movement which depends on the load condition, the wave behavior in the cruising area, the service speed, etc.; the natural frequency of the whipping fluid depends on the geometry of the tank, the filling level and the nonlinear effects of the large movement; The damping effects of the internal structure that may reduce the flapping motion to some extent; The viscosity of the fluid although there is no systematic research showing this yet; Gas effects [1]. The hydrodynamic pressure due to the pulse movement is divided into two parts: impulse pressure and non-impulsive pressure. Pulse pressure is the rapid pressure pulse when liquid hits a wall. Usually, it is the local high pressure within a short time. Impulse pressure is always related to moving waves and hydraulic jump, or very large standing waves. For non-impulse pressure, deceleration was the commonly observed variable pressure that is most related to the standing wave [2,3].

Fluid-structure coupling is one of the applications of multifold coupling problems in scientific and engineering problems. The governing equations of fluid-solid coupling are composed of governing equations of the flow field and structural field or other physical fields. There are two main methods of the numerical solution, including direct coupling solution and separation solution [4]. Hydroelectricity is related to the elastic bodies deformation replying to simultaneously and hydrodynamic excitations, modulation of this excitement due to deformation of the body. Associated fluid movement problems and elastomeric deformations of the body upon contact with a liquid are difficult to study in both cases: numerically and theoretically [5]. The present work is focused on studying the sloshing problem of tanks of rectangular cross section in 3D using FEM so as to predict the effect of the pressure of the fluid flowing out on the tank according to influencing of the levels of filling and baffles. The load generation was transferred to the structure analysis, which distributions incorporated to determine the

transient stress and deformation. The main outcome of this study is quite useful to understand the true physical phenomena of sloshing and design tank.

PHYSICAL MODEL

The current model involves a 3-D liquid storage tang of rectangular cross section, filled in a different level of water ($\mu = 0.00103$ kg/m.s, $\rho = 999.98$ kg/m^3). The dimensions of the tank are 1.2x0.5x0.2 m^3 and the water filling levels are 30, 50, and 70% with respect to the total tank height and other rest space is filled with air. Through the provocation process, model of the tank will undergo the effects of sloshing, which helps to create the pressure and forces on the tank wall. In order to minimize the sloshing effect on the wall of tank, baffle is equipped to the tank, with the dimensions of ($20x250x200\ mm^3$). The details of the physical model are presented in Fig. (1) [7,8].
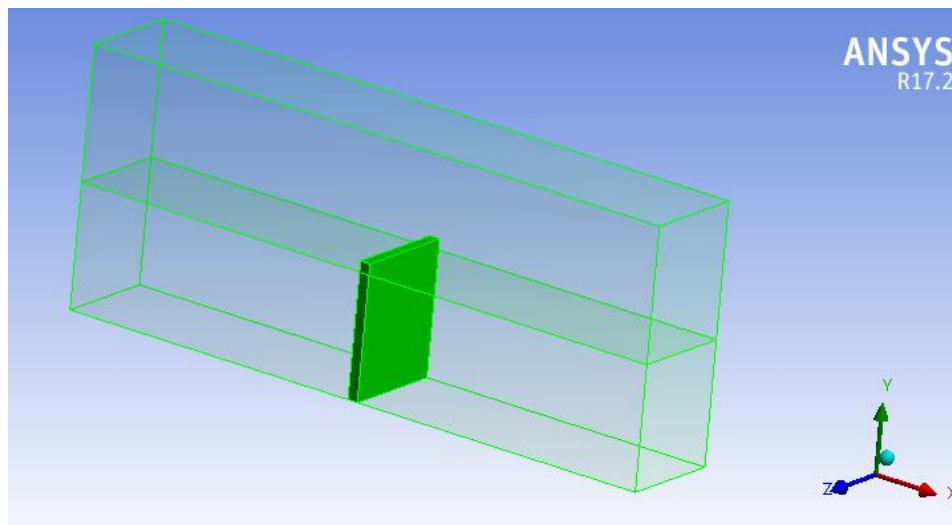

Figure 1. Liquid storage in rectangular tank capacity in 3D model.

INTERACTION OF FLUID STRUCTURE

Interaction of Fluid Structure deals with the conjugation of structural deformation and unsteady fluid flow as shown in Fig. (2). A two-way of Interaction of Fluid Structure was applied for disband solid and fluid parts. Therefore, the coupling of structure motion and fluid's pressure is important. The whole tank for sloshing action was a solid structure, which is impractical while the flexibility of the real tank structure must be considered. Thus, work on water resilience is needed to validate the research and improve the outcome [6].
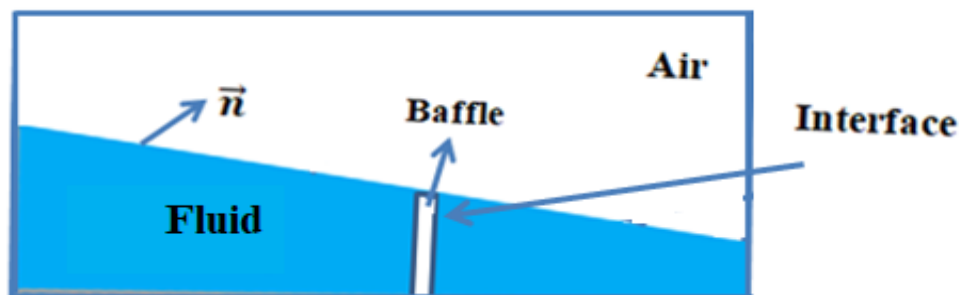

Figure 2. Sketch of sloshing in a rectangular storage tank.

MATHEMATICAL MODEL

The mathematical formulation and methods of solution, which used for studying of the sloshing are presented in this part. Equations of mathematical models describe the liquid flow in case of sloshing action including the free surface part. The Volume of Fluid (VOF) model is applied to track the interface of air-fuel and so the wave motion

action. The continuity, momentum, and phase equations are the governing equations which are presented, respectively [9, 10, 11]:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho v) = 0 \tag{1}$$

$$\frac{\partial}{\partial t}(\rho \bar{v}) + \nabla.(\rho \bar{v} \bar{v}) = -\nabla P + \nabla.\bar{\tau} + \rho \bar{g} + \bar{F} \tag{2}$$

$$\frac{D\alpha}{Dt} = \frac{\partial \alpha}{\partial t} + \nabla.(\alpha v) = 0 \tag{3}$$

Where; t is time; v is the fluid velocity vector, ρ is fluid density, α is volume fraction, τ is shear stress, and g is acceleration of gravity. The symbol used for terms of user defined source is F, for example the momentum source is produced by multiply of instantaneous acceleration and the density of a specific mesh cell.

The interface between the phases and free surface are tracked by shear stress. The density of two fluids can be symbolized by ρ_1 and ρ_2, and can be calculated using the volume fraction (α) equation:

$$\rho = \alpha \rho_1 + (1 - \alpha) \times \rho_2 \tag{4}$$

The dynamic viscosity of the two fluids is (μ_1, μ_2), and can be calculated using the volume fraction (α) equation:

$$\mu = \alpha \mu_1 + (1 - \alpha) \times \mu_2 \tag{5}$$

The liquid quality per unit volume of domain is termed as the phase function coefficient (α). If (α) is zero, this indicate that the cell is entirely filled with air, and if (α) is unity, this refer to that the cell is entirely filled with liquid. The behavior of the structure in response to fluid motion is determined by the two conservation principles of mass and momentum equation [12] as follows:

$$\nabla v^2 = 0 \tag{6}$$

$$\rho^s \left(\frac{d^2y}{dt^2} + \zeta \frac{dy}{dt} - f^s \right) - \nabla.\sigma^s = 0 \tag{7}$$

Where, the superscript (s) indicates the structure $v, \rho, y, \zeta, f \ and \ \sigma$ are the velocity, material density, structural displacement, damping coefficient, external force, and stress tensor respectively.

NUMERICAL MODEL

A compressible and incompressible fluid flows in 3D Reynolds Averaged Navier Stokes equation can be solved using ANSYS software, which is applied as the base solver. The tracking of free surface movements are analyzing and discretion the fluid governing equations using the Fluent base solver, which are solved by RANS and the aid of VOF. The pressure interpolation was tracked using the body force weight scheme, and the discretization of the momentum equation was achieved using the second order upward approach. The calculation of diffusion fluxes and convection through the control volume faces is done by using the Pressure Implicitly with Sloshing Operators (PISO) approach [13]. Grid and mesh were adopted using the dynamic mesh technique, which can be updated in every time step in solid and fluid sides, only the momentum function source was established into the x-momentum equations by using the UDF's in the Fluent software [14].

Fluent software imports the UDF to provide the standards functions. It is easy to specify the initial and boundary conditions, or properties of the material used in the case study. The language of C-programming is applied to write the UDF with the aids of visual expressions [15, 16]. The temporal grid size and grid size for the case of a no-slip boundary with the smooth boundary conditions are affected the computational results quality. The structural dynamics calculation uses the finite element method, which discretizes the calculation domain into finite element and solves the stiffness matrix at each node to obtain the solution, which has low grid requirements and fast convergence. The calculation is simple; therefore, the mechanicals own meshing module is used to discretize the solid computing domain.

Numerical Method Validation

The purpose of validation of numerical method is to ensure the capability of the CFD to predict the fluid-free surface behavior during the outer actions in the moving tank (container). The comparison between the CFD and experimental results using the model of sloshing mentioned in Ref. [17]. The fill depth of the tank in the experiment was fixed as 50%. The profile elevation of free surface liquid is numerically predicted for the same filling level. The time histories of free surface wave height of the tank wall are illustrated by figures (3) and (4). From the two figures, a good agreement between the experimental and CFD results. As a result, a good agreement is obtained between the CFD and experimental results.
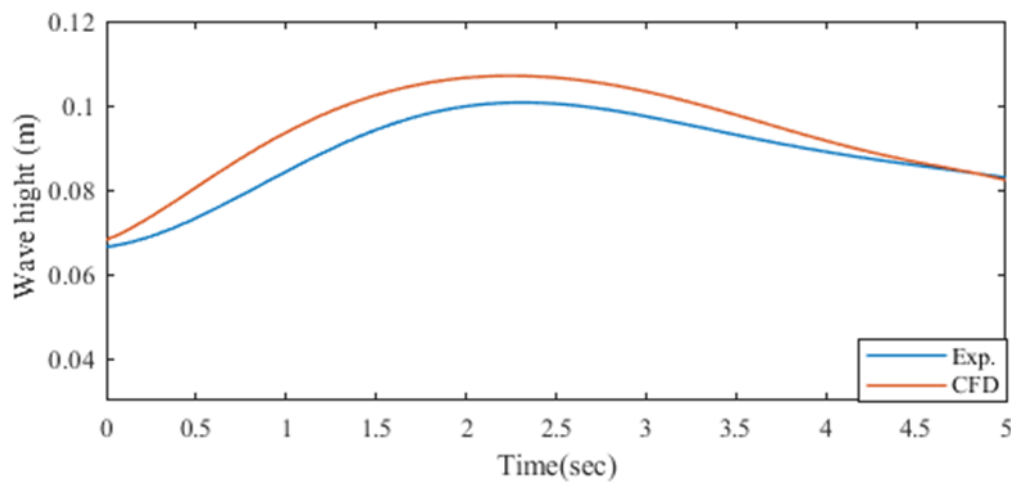

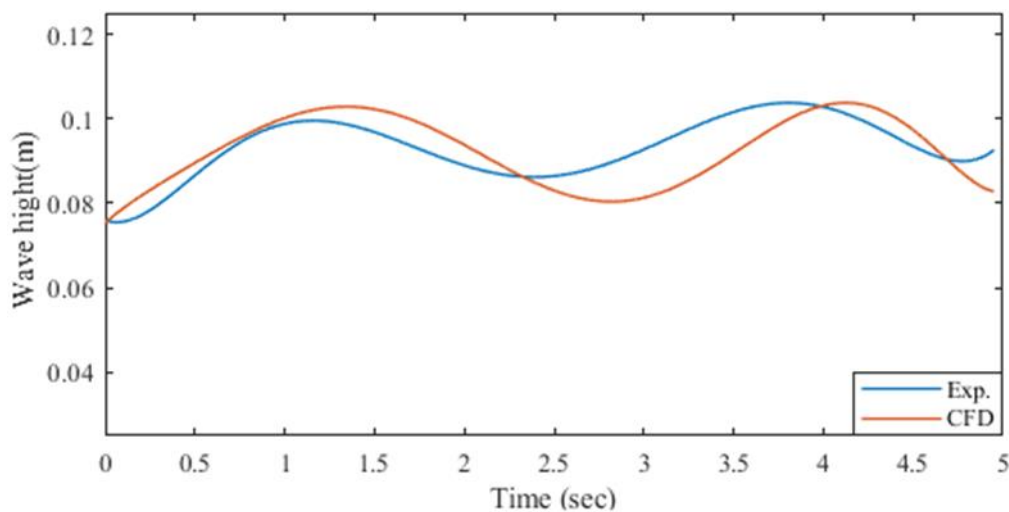**Figure 3.** Elevation of liquid free surface at (the right wall tank) without using baffled

Figure 4. Elevation of liquid free surface at (the right wall tank) using single baffle

RESULTS AND DISCUSSION

Results were obtained for the modelling of the interaction of fluid-structure in partly filled fluid tanks. The case study focuses on the phenomena of sloshing and the coupling of CFD analysis within the analysis of finite element, which is used to predict the pressure exerted on the baffle, the sloshing effect on the anchoring point forces, and the sloshing wave amplitude. The phenomena of coupling was adopted to simulate the impulsive pressure being amplified by the wall elasticity and sloshing effect. The presented results are in the forms of pressure distribution and graphs at the tank baffle with time (history). Simulation is carried out using ANSYS Fluent. The influence of liquid filling depths (30%, 50%, and 70%) on the tank system can be predicted on the free surface shape and estimated of the pressure effect on the tank wall. The analysed area is represented in following figures. Figure (5) reveals the free surface position during the excitation for different filling, which is reconstructed by the VOF

technique. The result shows the liquid is returned from the rear and front walls of the tank. For the case of a minimum liquid filling depth such as 30%, since there is less liquid, the liquid rushing out of the free surface during sloshing action is also higher. Some of the liquid promptly drops back to the free surface after awakening along the wall of the tank [18, 19]. Since the rising height of liquid is not high, the liquid will return to the original free surface. The impact on the sidewall of the tank is small, because the pressure from the surface is not large. Thus, the riskiness of the sloshing action is not considered.
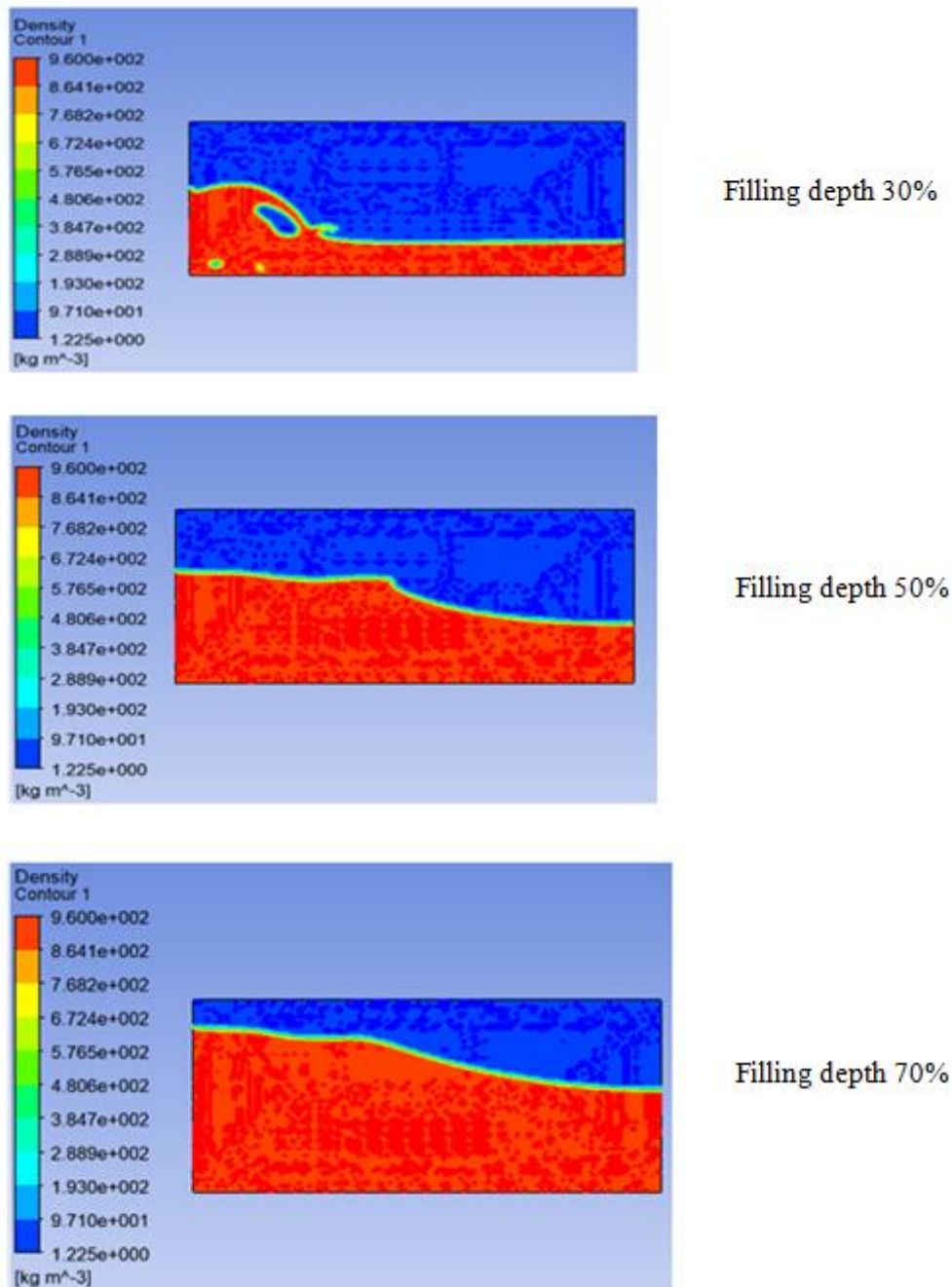
Figure 5. Free surface elevation for liquid at different filling depth on the tank.

For the case of minimum filling level (50%), The increased height of the tank bulkhead is large, which leads to more impact on the tank top. Thus, it takes more time period for the liquid to be raised from the free surface to the top of liquid tank and the down back, causing relatively high slap pressure. Another case of filling liquid level (70%), since this level is near the top of the tank, the liquid rises a short distance along the tank bulkhead and impact the top of the tank, also the liquid hitting the sidewall with low effect. The drastic of sloshing not only causes sloshing waves to the liquid with high wave, but also may causes a jumps or vortices to the liquid. The

increasing of filling level decreases the free surface level and achieve a stable position [20, 21]. This due to the increase in the liquid mass flow rate will enhances the assimilation of the abrupt disturbance located to the tank wall. The highest value of pressure acting on the wall of tank will increases with increasing liquid volume in the tank. This behaviour is clearly presented in Fig. (6). This is harmonious with anticipation. The pressure fluctuation and the effect of the viscous damping decrease as the fill depth increased.
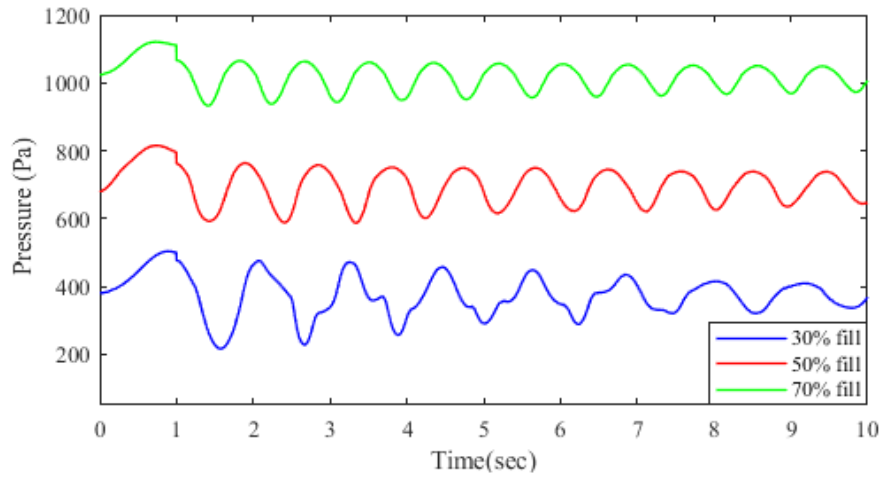

**Figure 6.** Comparison of the time histories of the pressure at tank for different filling depth.

In order to reduce the liquid sloshing in partially filled tank, baffles will be used to suppress the motion of the wave and absorb the kinetic energy of the liquid. A certain type of vertical baffle is located in the partially filled level that affects the elevations of the liquid free surface [22, 23]. The pressure acting on the wall of tank are presented in figures (7) and (8), respectively.
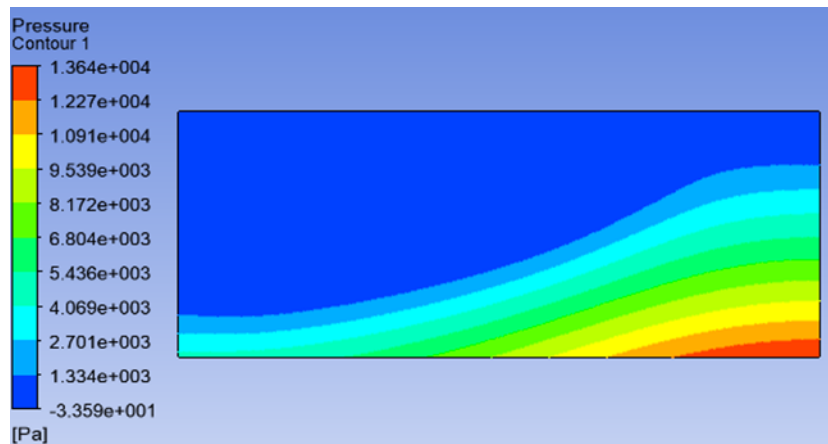
a.   Without baffled at filling 50%.
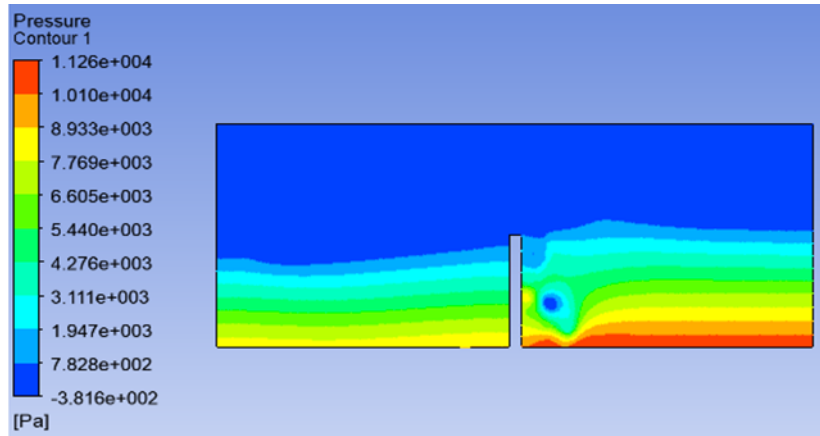
b.   With single baffled at filling 50%.
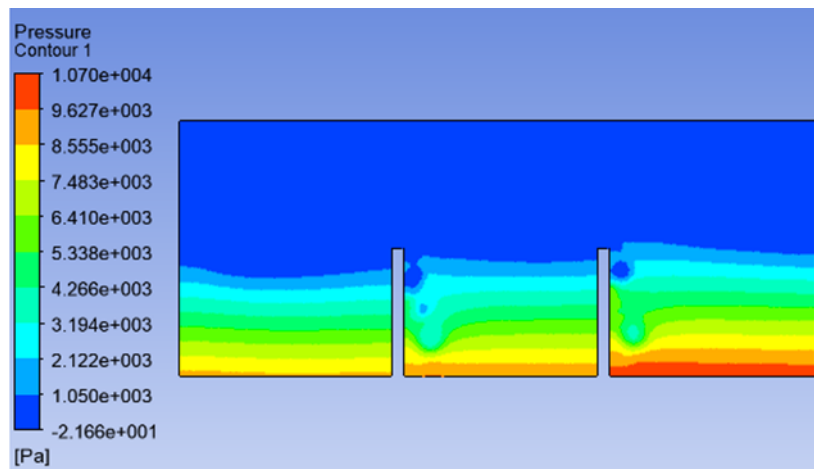
c.   With two baffled at filling 50%.

Figure 7. Pressure distribution contour in a baffle and unbaffled tank during excitation.

It is obviously shown that the motion of the free surface waves are restricted by the use of baffle and the characteristics length was reduced, which will reduces the amplitude of the free surface motion [24]. The used vertical baffles reduce the maximum impact pressure by about 20%. baffles can cause turbulence and vortex due to their sharp edges that violent energy which can fall back, so the fluid is unable to ascend. The increasing in the baffle number can reduces the pressure in the side of tank wall (sloshing motion).
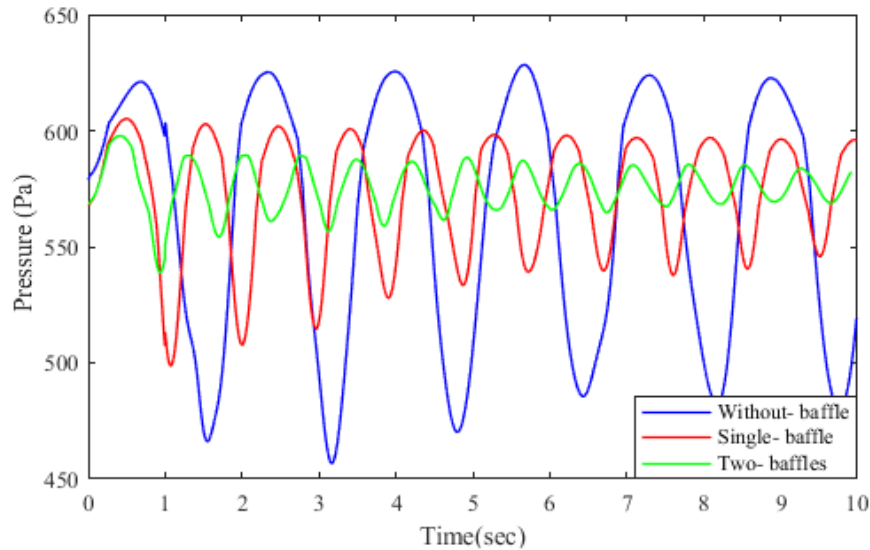
Figure 8. Comparison of the time histories of the pressure for the baffled and unbaffled tank.

Fluid structure Interaction Analysis

The elastic body analysis technique described in the section was applied to the deflection problem of 3D transient modeling of the sloshing effect using the Ansys program. The application of fluid-structure interaction can be presented using coupling method with the aid of two-way FSI coupling method, which is efficient to predict the pressure acting on the structure in various ways. The preliminary design of the baffling material against the load is assumed from the Aluminium alloy [15]. The results reveal that the stress and deformation in the baffled increased with increasing filling level. It was also proven that a high pressure area was generated on the front baffle surface. The maximum deformation and maximum stress of the baffled tank have similar behaviour. Figure (9) shows the maximum stress of baffle changing with time history and it happens obviously on the baffle bottom. The simulation results show the maximum value of the transient stress at the fixed end compared to the analysis solution and the tendency to converge to zero at it goes to the free end of the baffle.
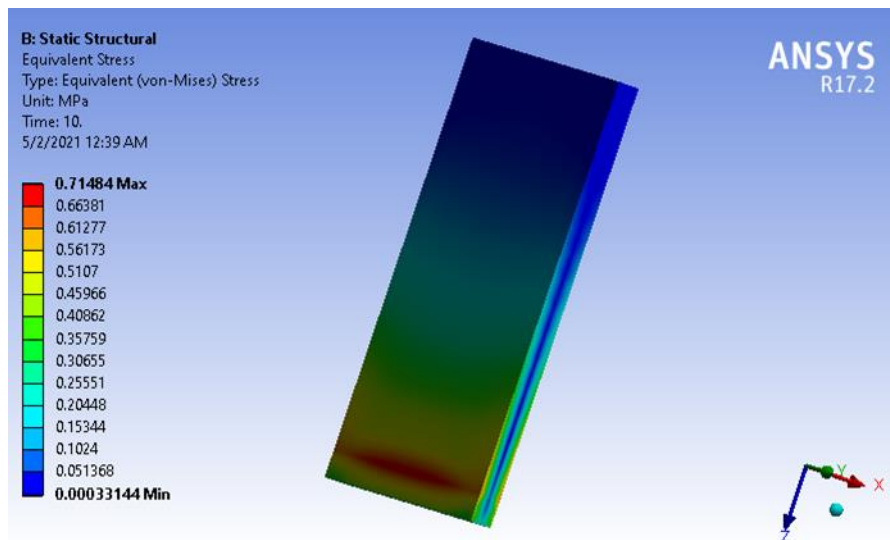

Figure 9. Equivalent stress contour for a baffled tank at 50% filling depth.

The transient stress of the baffled was illustrated for three different filling depths (30%, 50%, and 70%) as was shown in Fig. (10). The maximum deformation and maximum stress happened at the maximum filling level because of a high pressure will be generated on the front surface. The maximum value of Von Mises stress (1.2Mpa) was found at the root of the baffle relative to the pressure that occurs on the tank wall. Also, the

behaviours of the maximum stress were adapted according to the pressure. The maximum value of transient stress is increased about 55% from the other, which can be conceded for the design of the tank structure.
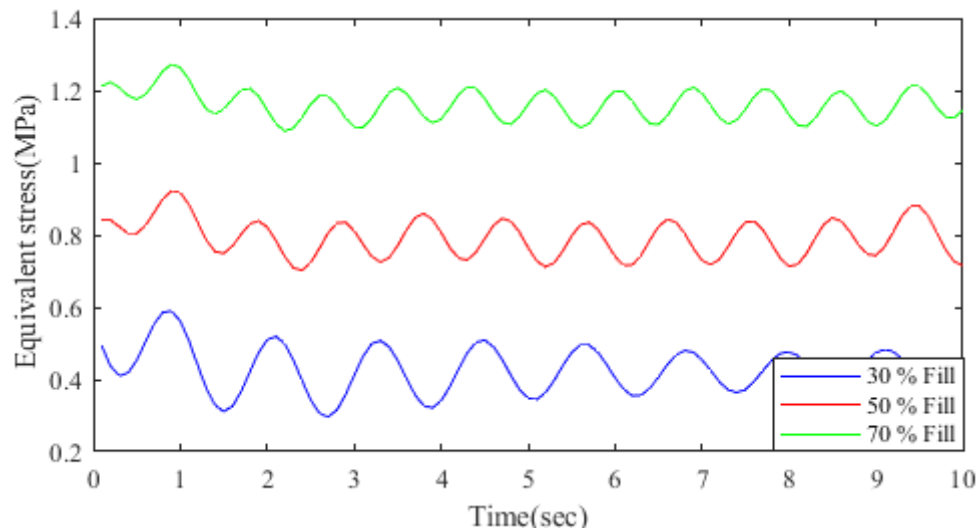

Figure 10. Equivalent stress of baffled that changes by the time with different filling depth.

Figure (11) shows the maximum deflection occurs at the free side of the tip baffle, while the deflection of the baffled is proportional to the pressure distribution over the baffle and inversely proportional to the stiffness factor EI, modulus of elasticity, and second-moment area. It is clearly seen that the deformation was increased as a result of the pressure distribution over the baffled tank.
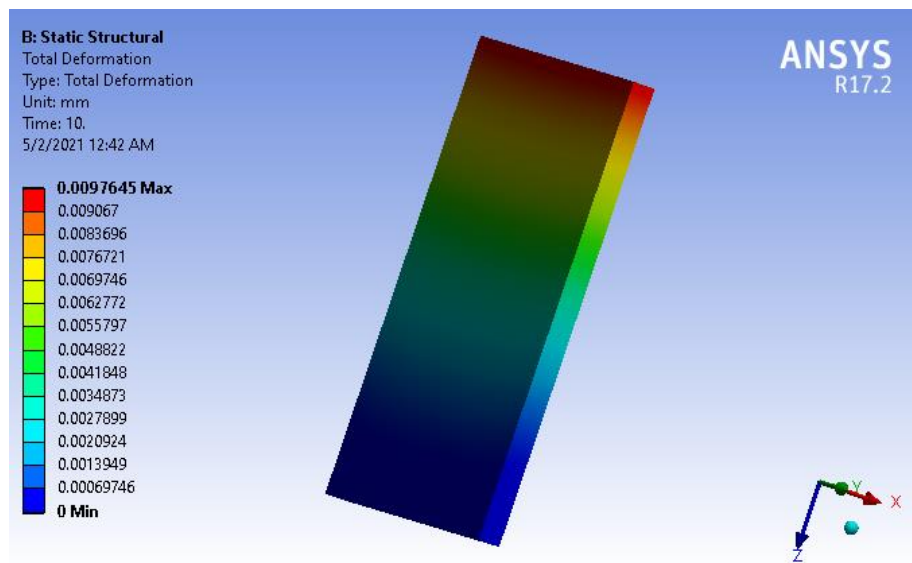

Fig 11. Total deformation contour for a baffled tank at 50% filling depth.

Moreover, it was found that the larger increment of deformation obtained with filling depth and the maximum value of deformation is greater about 55% of the other cases, the maximum deflection occurs at the free side of the baffled on the tip at about (0.016) mm.
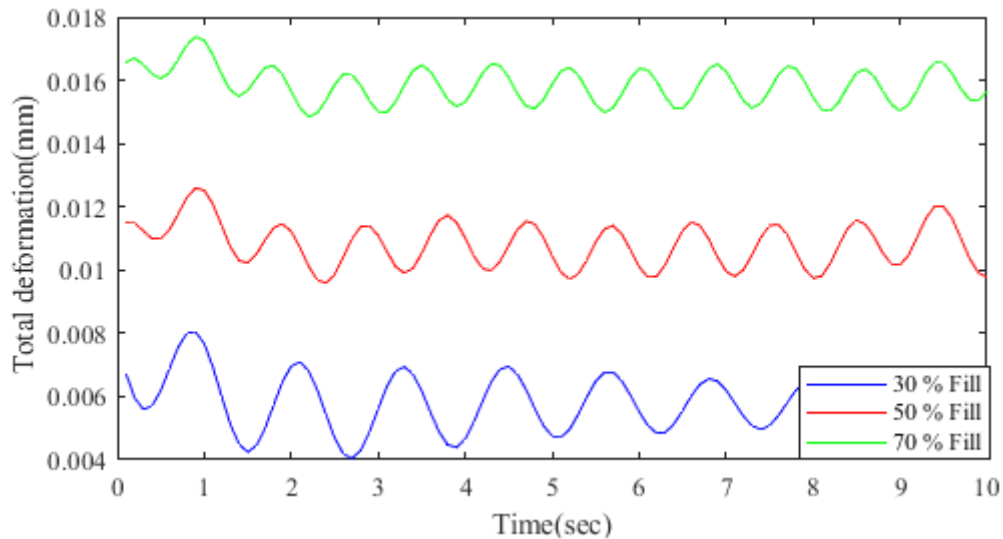
Figure 12. Total deformation variation for a baffled tank at different filling depths.

The equivalent transient stress-induced over the structure with and without baffled at filling depth 50% as shown in Fig. (13). Based on the results presented, the larger diminished equivalent stress was obtained at baffled. The maximum value of transient stress is decreased about 10% from the other, which can be conceded for the design of the tank structure.
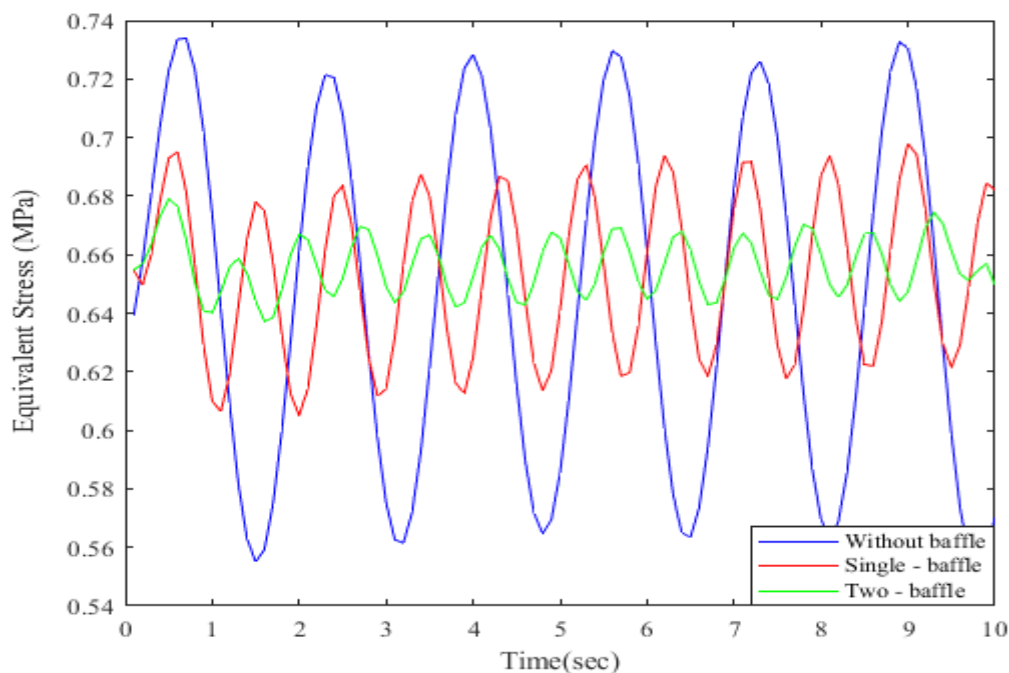

Figure 13. Equivalent stress variation in a baffled and unbaffled tank during excitation.

Figure (14) was displayed the deformation with and without baffle at filling depth 50%, the maximum deflection occurs at the free side of the tip baffle, while the deflection of the baffle is proportional to the pressure distribution over the baffle wall. It is clearly seen that the deformation was increased as a result of the pressure distribution over the baffle. Moreover, it was found the larger increment of deformation was obtained without baffle and the maximum value of deformation is greater by nearly 10% of the other, the maximum deflection occurs at the free side of the baffle on the tip.
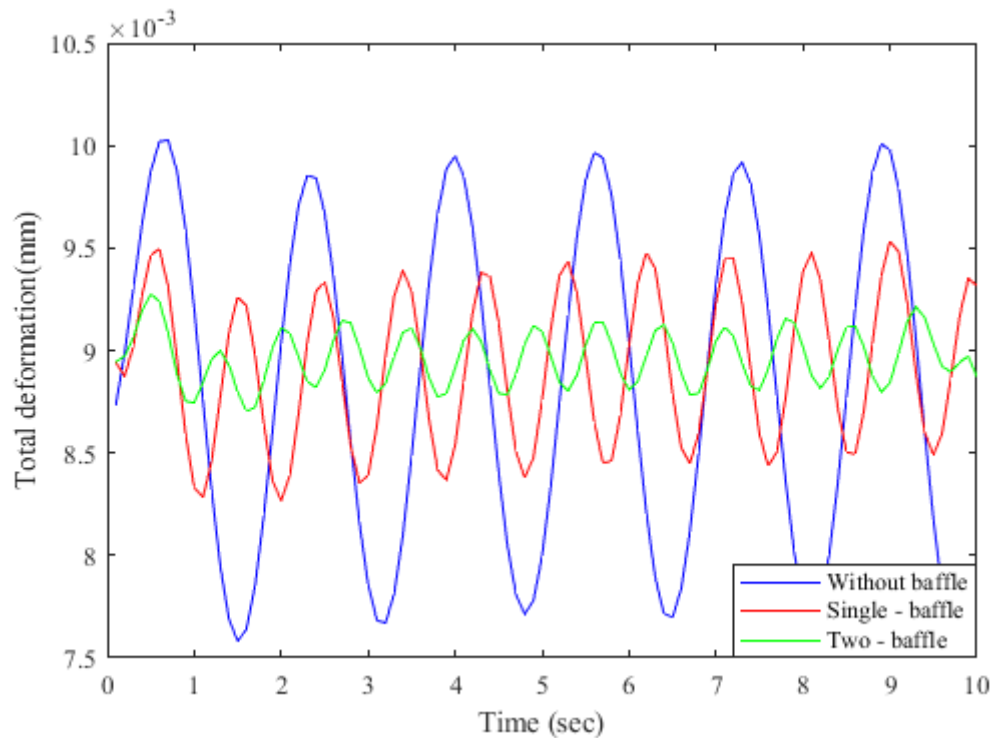
Figure 14. Total deformation in a baffled and unbaffled tank during excitation.

CONCLUSIONS

This study is based on the self-developed particle approach to predict the liquid sloshing problem in rectangular tanks under outer excitations. A system coupling was applied for combining the solid and fluid domain. The dynamic mesh technique was applied for regenerating the mesh, after the effectiveness of the 3D numerical tank with baffles; the resulting stresses and deformations were calculated for different filling rates. It was found that the baffles prevented the fluid rising towards the tank wall and therefore reduced the effects of sloshing that without a doubt will significantly increase the life of the structural elements. Moreover, the increase in the baffle number can minimize the sloshing or pressure actions, resulting in reducing the transient stresses and the deformation of the tank.

REFERENCES

[1] H. Akyildiz and N. Erdem Unal, "Sloshing in a Three –Dimensional Rectangular Tank: Numerical Simulation and Experimental Valuation", Ocean Engineering, 2006.

[2] R.A. Ibrahim, "Liquid Sloshing Dynamics Theory and Applications", Book, University of Cambridge, 2005.

[3] E.Q. Hussein, B.R. Sadeq, and F.L. Rashid, "Failure Analysis of Composites Plate with Central Opening Hole Subject to Arbitrary Tension Load", Journal of Mechanical Engineering Research and Developments, Vol. 43, No. 1, Pp. 36-47, 2020.

[4] S. Magadlna and P. Janusz, "Numerical Analysis of Fluid Inside Partially Filled Container which is moving in unsteady way", Institute of Aeronautics and Applied Mechanics, 2015.

[5] Y.M. Shen, C.O. Ng, and Y.H. Zheng, "Simulation of Wave Propagation Over a Submerged Bar Using the VOF Method with a Two-Equation K-e Turbulence Modeling", Ocean Engineering, 2004.

[6] L. Hou, and F. Li. "A Numerical Study of Liquid Sloshing in Two- Dimensional Tank Under External Excitation", Journal of Marine Science, 2012.

[7] Z. Haitao, and B. Sun, "Numerical Simulation of Sloshing in 2D Rectangular Tanks Based on the Prediction of Free Surface", Mathematical Problems in Engineering, 2014.

[8] M. Asaduzzaman, "Interfacial Wave Dynamics of Core-Annular Flow of Two Fluids", University of Pennsylvania, Doctor, 2018.

[9] E.Q. Hussein, F.L. Rashid, and H.N. Azziz, "Aerodynamic Heating Distribution for Temperature Prediction of Fast Flying Body Nose Using CFD", Journal of Advanced Research in Fluid Mechanics and Thermal Sciences, Vol. 64, No. 2, Pp. 21-27, 2019.

[10] B. Godderidge, An Investigation of Multiphase CFD Modeling of a Lateral Sloshing Tank, 2009.

[11] D. HJ Al-Zubaidi, M. Al-Waily, E.Q. Hussein, "Analytical Investigation of Crack Depth and Position Effect onto Beam Force Vibration Response with Various Harmonic Frequency Influence", International Journal of Energy and Environment, Vol. 11, No. 1, Pp. 1-28, 2020.

[12] H.J. Diyaa, Al-Zubaidi, M. Al-Waily, E.Q. Hussein, M.A.R. Sadiq Al-Baghdadi, "Experimental and Numerical Temperature Distribution Study for Harmonic Vibration Beam with and without Crack Effect", International Journal of Energy and Environment, Vol. 10, No. 6, Pp. 373-384, 2020.

[13] R. Suyal. "CFD Analysis of Fuel Sloshing in a Cylindrical Tank with and Without Baffle Under Linear Acceleration", National Institute of Technology, Rourkela, Thesis, 2016.

[14] G. Bernhard, "An Investigation of Multiphase CFD Modeling of a Lateral Sloshing Tank", Computers & Fluids, Vol. 38, No. 2, Pp. 183-193, 2009.

[15] Fluent Inc., FLUENT 6.0 User's Guide: vol. 5, USA, 2001.

[16] ANSYS Inc. Fluent, 2010: http://www.ansys.com/products/fluid-dynamics/fluent/. Access 10\05\2021.

[17] R. Rajagounder, G.V. Mohanasundaram, and P. Kalakkath, "A Study of Liquid Sloshing in an Automotive Fuel Tank under Uniform Acceleration", Engineering Journal, Vol. 20, No. 1, 2015.

[18] H.B. Mahood, A.S. Baqi, and F.L. Rasheed, "Two-Phase Pressure Drop Through Obstructions", Iraqi Journal of Chemical and Petroleum Engineering, Vol. 4, No. 2, Pp. 49-55, 2003.

[19] G.M. Taih, K.R. Azeez, and R.F. Leftah, "Using water energy for electrical energy conservation by building of micro hydroelectric generators on the water pipelines that depend on the difference in elevation", Energy, Power and Control (EPC-IQ), 1st International Conference, IEEE, Pp. 379-383, 2010.

[20] I.K. Fayyadh, F.L. Rashid, F.A. Jasim, and A. Hashim, "Calculation of Drag Coefficients for Geometrical Shapes in Gas Using Numerically Transport Equation solution", British Journal of Science, Vol. 3, No. 2, Pp. 63-79, 2012.

[21] M.I. Ismael, F.G. Ibrahim, and R.F. Lafta, "Investigation of Townsend Energy Factor and Drift Velocity for Air in Terms of L, G & E/P", Journal of University of Babylon, Vol. 21, No. 5, Pp. 1598-1610, 2013.

[22] A.A. Abd, S.Z. Naji, and F.L. Rashid, "Efficient Design of a Large Storage Tank for Liquefied Natural Gas", Journal of University of Babylon for Engineering Sciences, Vol. 26, No. 6, Pp. 362-383, 2018.

[23] E.Q. Hussein, B.R. Sadeq, and F.L. Rashid, "Fluid Structure Interaction of Non–Return Valve Using CFD Technique", Journal of Mechanical Engineering Research and Developments, Vol. 43, No. 7, Pp. 137-148, 2020.

[24] A.I. Al-Zughaibi, E.Q. Hussein and F.L. Rashid, "A dynamic meshing technique for analysis of nonlinear sloshing in liquid tanks", IOP Conf. Series: Materials Science and Engineering, Pp. 1067, 2021. doi:10.1088/1757-899X/1067/1/012094