JOURNAL OF SUBSEA AND OFFSHORE

SCIENCE AND ENGINEERING

VOLUME.2

JUNE 20, 2015

ISSN: 2442-6415

PUBLISHED BY ISOMAse

Journal of Subsea and Offshore
-Science and Engineering-

Vol.2: June 20, 2015

Contents

ISOMAse

International Society of Ocean, Mechanical and Aerospace
-Scientists and Engineers-

Journal of Subsea and Offshore
-Science and Engineering-

Vol.2: June 20, 2015

About JSOse

The **Journal of Subsea and Offshore -science and engineering- (JSOse),** ISSN's registration no: 2442-6415 is an online professional journal which is published by the International Society of Ocean, Mechanical and Aerospace -scientists and engineers- (ISOMAse), Insya Allah, four volumes in a year which are March, June, September and December.

The mission of the JSOse is to foster free and extremely rapid scientific communication across the world wide community. The JSOse is an original and peer review article that advance the understanding of both science and engineering and its application to the solution of challenges and complex problems in subsea science, engineering and technology.

The JSOse is particularly concerned with the demonstration of applied science and innovative engineering solutions to solve specific subsea and offshore industrial problems. Original contributions providing insight into the use of computational fluid dynamic, heat transfer, thermodynamics, experimental and analytical, application of finite element on offshore and subsea, offshore structural and impact mechanics, stress and strain localization and globalization, metal forming, behaviour and application of advanced materials in shallow and deepwater, shallow and deepwater installation challenges, vortex shedding, vortex induced vibration and motion, flow assurance, ultra-deepwater drilling riser, wellhead integrity and soon from the core of the journal contents are encouraged.

Articles preferably should focus on the following aspects: new methods or theory or philosophy innovative practices, critical survey or analysis of a subject or topic, new or latest research findings and critical review or evaluation of new discoveries.

The authors are required to confirm that their paper has not been submitted to any other journal in English or any other languages.

ISOMAse

International Society of Ocean, Mechanical and Aerospace
-Scientists and Engineers-

Scope of JSOse

JSOse welcomes manuscript submissions from academicians, scholars, and practitioners for possible publication from all over the world that meets the general criteria of significance and educational excellence. The scope of the journal is as follows:

- Shallow, Deep and Ultra-deep Water and Arctic Pipelines and Offshore Structure
- Shallow, Deep, Ultra-deep Water Installation Challenges
- Subsea and Offshore Challenges in Pipe Materials, Flow Assurance, Multi Phases Flow, Equipment and Hardware
- Flexible Pipe and Umbilical
- Riser, Mooring Lines Design and Mechanics System
- Advanced Engineering, Lateral and Upheaval Buckling, Pipeline Soil Interactions
- Challenges of High Pressure - High Temperature (HPHT) in Ultra-deep Water
- Vortex Shedding and Vibration Suppression (VIV & VIM)
- Project Experiences, Case Study and Lessons Learned on Subsea and Offshore
- Integration of Management, Materials, Safety and Reliability
- Certified Verification Analysis (CVA)
- Subsea and Offshore Structures Construction.

The International Society of Ocean, Mechanical and Aerospace –science and engineering is inviting you to submit your manuscript(s) to isomase.org@gmail.com for publication. Our objective is to inform the authors of the decision on their manuscript(s) within 2 weeks of submission. Following acceptance, a paper will normally be published in the next online issue.

Journal of Subsea and Offshore
-Science and Engineering-
Vol.2: June 2015

Editors

Chief-in-Editor

Jaswar Koto	(Ocean and Aerospace Research Institute, Indonesia) (Universiti Teknologi Malaysia, Malaysia)

Associate Editors

Ab. Saman bin Abd. Kader	(Universiti Teknologi Malaysia, Malaysia)
Adhy Prayitno	(Universitas Riau, Indonesia)
Adi Maimun bin Abdul Malik	(Universiti Teknologi Malaysia, Malaysia)
Ahmad Fitriadhy	(Universiti Malaysia Terengganu, Malaysia)
Ahmad Zubaydi	(Institut Teknologi Sepuluh Nopember, Indonesia)
Bambang Purwohadi	(The Institution of Engineers Indonesia, Indonesia)
Carlos Guedes Soares	(University of Lisbon, Portugal)
Harifuddin	(DNV, Batam, Indonesia)
Hassan Abyn	(Persian Gulf University, Iran)
Mohamed Kotb	(Alexandria University, Egypt)
Moh Hafidz Efendy	(PT McDermott, Indonesia)
Mohd Nasir Tamin	(Universiti Teknologi Malaysia, Malaysia)
Mohd Yazid bin Yahya	(Universiti Teknologi Malaysia, Malaysia)
Mohd Zaidi Jaafar	(Universiti Teknologi Malaysia, Malaysia)
Musa Mailah	(Universiti Teknologi Malaysia, Malaysia)
Priyono Sutikno	(Institut Teknologi Bandung, Indonesia)
Radzuan Junin	(Universiti Teknologi Malaysia, Malaysia)
Rudi Walujo Prastianto	(Institut Teknologi Sepuluh Nopember, Indonesia)
Sergey Antonenko	(Far Eastern Federal University, Russia)
Sunaryo	(Universitas Indonesia, Indonesia)
Sutopo	(PT Saipem, Indonesia)
Tay Cho Jui	(National University of Singapore, Singapore)
Wan Rosli Wan Sulaiman	(Universiti Teknologi Malaysia, Malaysia)

ISOMAse

International Society of Ocean, Mechanical and Aerospace
-Scientists and Engineers-

Journal of Subsea and Offshore
-Science and Engineering-, Vol.2

June 20, 2015

CFD Simulation for Stratified Oil-Water Two-Phase Flow in a Horizontal Pipe

Adib Zulhilmi Mohd Alias,[a] J.Koto,[a,b,*] , and Yasser Mohamed Ahmed,[a,c]

[a)] *Department of Aeronautics, Automotive and Ocean Engineering, Universiti Teknologi Malaysia, Malaysia*
[b)] *Ocean and Aerospace Research Institute, Indonesia*
[c)] *Dept. of Naval Architecture and Marine Engineering, Faculty of Engineering, Alexandria University, Alexandria, Egypt*

*Corresponding author: jaswar@mail.fkm.utm.my and jaswar.koto@gmail.com

Paper History

Received: 4-June-2015
Received in revised form: 15-June-2015
Accepted: 19-June-2015

ABSTRACT

Oil-water two-phase flow in 0.0254m horizontal pipe is simulated using FLUENT 6.2. The stratified flow regime is modeled using Volume of Fluid (VOF) with turbulent model RNG k-ε. Grid independent study has been conducted to decide mesh size for solution accuracy and optimum computational cost. The simulation is performed in time-dependent simulation where oil and water are initially separated by patching the region base on difference in density. Observation on the effect of velocity to the pressure gradient was also simulated. Flow velocity at 0.2, 0.5, 0.8 and 1.1 m/s with same volume fraction for each phase with appropriate multiphase model and turbulence model are presented.

KEY WORDS: *Stratified oil-water flow; Turbulence flow; CFD*

1.0 INTRODUCTION

Immiscible liquid-liquid flow is a common occurrence encountered in a variety of industrial processes. In oil and gas industry, oil transportation either from reservoir to processing facilities or to onshore refinery are usually transported in multiphase flow condition since water and oil are normally produced together. Fractions of water are usually influenced by its existence within the stratum and also through oil recovery method which used water to enhance the remaining oil in the reservoir.

The presence of water, during the transportation of oil has a significant effect because the flow is no longer can be treated as a single-phase flow. Oil-water has complex interfacial structure which complicates the hydrodynamic prediction of the fluid flow. Changes in water fraction may influence the power required to pump the fluid due to corresponding changes in pipeline pressure drop. Either water-in-oil or oil-in-water dispersions, both can influence the pressure gradient dramatically.

Computational fluid dynamics (CFD) techniques have been used to simulate the stratified pipe flow. One of the early CFD models of turbulent stratified flow in a horizontal pipe was presented by Shoham and Taitel [1] where a 2D simulation for liquid-gas flow was simulated by adopting zero-equation models for the liquid region flow field while the gas region was treated as a bulk flow. Issa [2] numerically simulated the stratified gas-liquid pipe flow, using standard k-ε turbulence model with wall functions for each phase. Newton and Behnia [3] obtained more satisfactory solutions for stratified pipe flow by employing a low Reynolds number turbulent model instead of wall functions.

Hui et al [4] simulated stratified oil-water two-phase turbulent flow in a horizontal tube by applying RNG k-ε model combined with a near-wall low-Re turbulence model to each phase and they adopt continuum surface force approximation for the calculation of surface tension. Their simulation results was compared with Elseth et al [5] who simulated the turbulent stratified flow, however their numerical results are not acceptable when compared with their measured data.

Stratified oil-water two-phase pipe flow was investigated using different type of multiphase model. Awal et al [6] achieved CFD simulation tool to investigate inline oil and water separation characteristics under downhole conditions. They chose the Eulerian-Eulerian model, which is computationally most comprehensive but more suitable for multiphase systems with the dispersed phase exceeding 10% v/v/. Carlos F. [7] developed a 2D model for fully-developed, turbulent-turbulent oil-water stratified flow using finite-volume method in a bipolar coordinate system and applying a simple mixing-length turbulence model. Hui et al [4] and Al-Yaari et al [8] simulated stratified oil-water two-phase turbulent flow in a horizontal tube numerically using a volume of fluid (VOF) model. They applied RNG k-ε model with enhanced wall function combined with optimum meshes through grid independent study to obtain clearly separated oil layer and optimum computational cost.

In the present paper multiphase model of Volume of Fluid (VOFis used to model the stratified oil-water flow. Optimum number of elements for simulation accuracy has been conducted through grid independent study. Observation on the effect of velocity to the pressure gradient was also simulated at flow velocity 0.2, 0.5, 0.8 and 1.1 m/s with same volume fraction for each phase.

2.0 NUMERICAL SIMULATION

2.1 Geometry and mesh

The domain and the meshes were created using ANSYS Design Modeler. A sketch of the geometry of the calculation domain is shown in Figure 1. The geometry consists of semicircular inlet for oil and water with 1 meter length of the flow domain. The inlet for both phases is at the same inlet face where oil on top and water at the bottom region. This will initially made the flow in stratified condition. In addition, as both inlets also flew with a same velocity with direction almost parallel to each phase makes fewer disturbances to maintain stratified flow. The diameter of the pipe for the present work is 0.0254 m. In order to keep the volume of oil and water are flowing continuously throughout the domain until the outlet, patch file and adapt region is used to declare the top and bottom regions for oil and water. This will avoid insufficient volume of either phase.

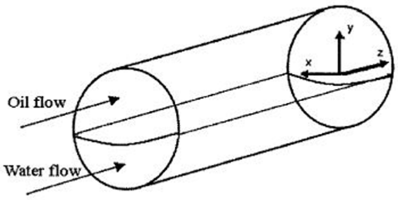


Figure 1: Schematic representation of pipe flow

A block-structured meshing approach was used to create meshes with only tri/tet cells. To obtain fine meshing scheme, sizing was setup with curvature normal angle ll degree, 0.0001 minimum size and 3.0 m maximum size. While to improve the flow near the wall region, two layer inflation with growth rate 1.2 is adapted

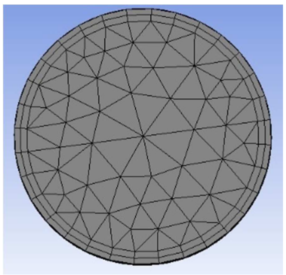


Figure 2: Tri/Tet meshes

2.2 Boundary conditions

There are three faces bounding the calculation domain: the inlet boundary, the wall boundary and the outlet boundary. Flat velocity profile for oil and water were introduced at the inlet of their sections. The outlet boundary condition at the end was set up as a pressure outlet boundary. No slip was used to model liquid velocity at the wall. The main fluid phases' physical properties are reported in Table 1.

Table 1: Fluid phases physical properties

Property	Water Phase	Oil Phase
Density (ρ), kg/m^3	998.2	780
Dynamic Viscosity (μ), Pa.s	0.001003	0.00157
Interfacial Tension, N/m	0.17 m @ 20^0C	

2.3 Solution strategy and convergence

Pressure-based solver is chose since it was applicable for wide range of flow regimes from low speed incompressible flow to high speed compressible flow. This solver also requires less memory (storage) and allows flexibility in the solution procedure. Green-gauss Node-Based is elected for higher order discretization scheme since it is more accurate for tri/tet meshes. For pressure, PRESTO! discretization scheme was used for pressure, second order upwind discretization scheme was used for the momentum equation, volume fraction, turbulent, kinetic and turbulent dissipation energy. Second-order upwind is chose rather than First-order upwind because it uses larger stencils for 2nd order accuracy and essential with tri/tet mesh even though the solution to converge may be slower but manageable. In addition, the simulation is time dependent (transient) with 1000 time steps, 0.01 time step size and 200 iterations at each time step size.

3.0 RESULTS

In this section one presents, use of Volume of Fluid multiphase model along with RNG k-ε for turbulent model, grid independent test and sample of pressure drop prediction using this simulation

.

3.1 Grid independent study

A grid independent study is conducted to obtain sufficient mesh density as it was necessary to resolve accurate flow. A grid independent solution exists when the solution does not change when the mesh is refined. The computational grid of 46631, 79488, 104584 and 142374 elements were tested for the mesh independent study to find out the optimum size of the mesh to be used for simulation. Figure 3 shows an oil volume fraction contours at plane z = 0.5 m which indicates the accuracy of the mesh to display the flow pattern. As shown in figure, system increased number of elements shows better prediction for stratified flow pattern with smoothness of the clearly oil and mixed layer. 46631 showing bad prediction on the oil and mixed layer since insufficient amount of elements could not give detail prediction especially on the mixed layer. Both meshes for 104584 and 142374 gave almost similar contours of oil fraction with slight differences in the smoothness of the clearly oil and mixed layer. Therefore, based on the oil volume fraction contours results, 142374 cells are the most optimum number of cells required to predict the oil-water stratified flow in the tested domain and such mesh is going to be used for simulation.

In addition, such decision has been tested by comparing the pressure profiles obtain for every mesh tested as shown in Figure 4. At mesh size 46631, 68204 and 79488, the pressure plot is away from the other plots. The pressure profile starts to unchanged with mesh 92440 until 171393. Before deciding the best meshes size, simulation cost also is required to look at. Since increase number of meshes will increase the amount of time for simulation, the meshes size of 142374 is the most optimum number of elements could be chose.

3.2 Pressure prediction at different flow velocity

By using the simulated oil-water stratified flow, pressure prediction at different flow velocity have been conducted. Flow velocity of 0.2, 0.5, 0.8 and 1.1 m/s with (0.5 input water volume fraction) as a sample flow pattern has been simulated. Volume of fluid (VOF) multiphase model with RNG k-ε model was used for simulation the tested domain containing 142374 cells (the optimum mesh size) based on the decision mentioned earlier in this paper. At such condition, the oil-water flow pattern simulated is seen stratified as shown Figure 5, with multiple layers of phase density in the middle of the pipe where the oil and water phases met. Figure 6 shows the view of oil volume fraction contours at pipe length (z = 0.5 m) which located in the middle of the pipe length. Different velocity indicates different inversion point. 0.2 and 0.5 m/s can be considered as slow speed which gives more time for both phases to dispersed within each other. On the view of oil production is not good since avoiding mixing phases will reduce time during separation processes. 0.8 and 1.1 m/s shows better oil and water mixture. From the contours seen the fraction of oil at the upper region shows high fraction of oil. This indicates less water inversion to its phase.

4.0 CONCLUSIONS

The following conclusive remarks result from our analysis. As far as the fluid dynamic analysis is concerned:

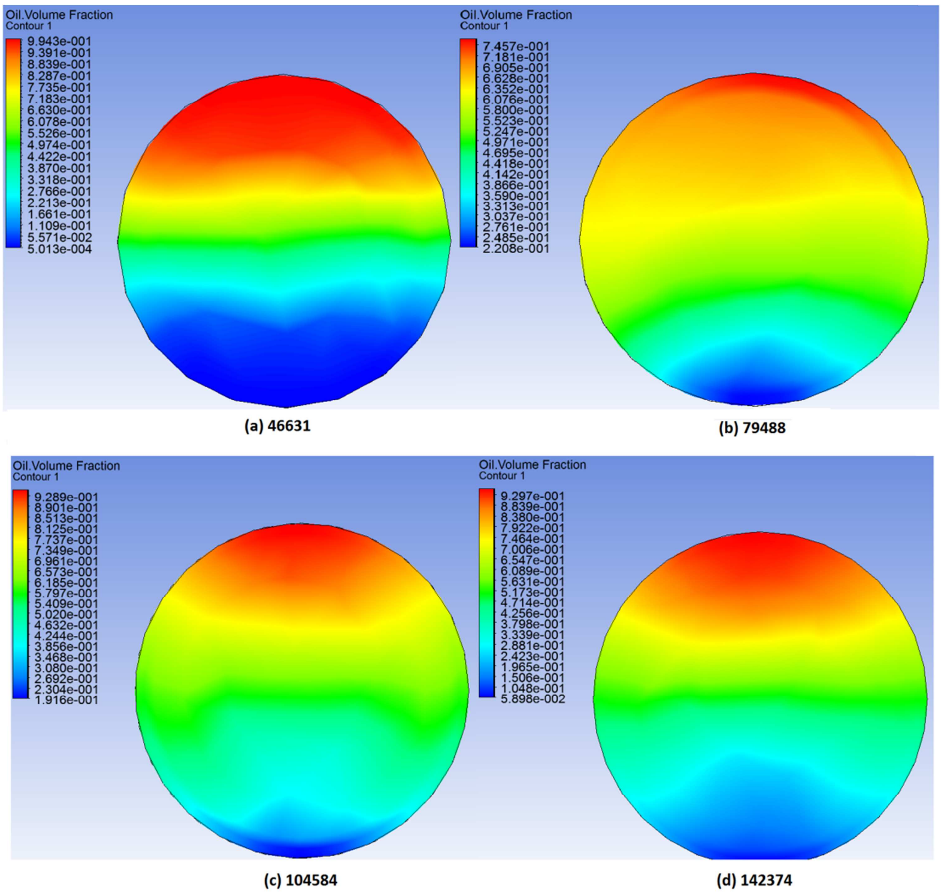
1. CFD calculations using Fluent 6.2 were performed to predict the oil-water stratified flow in 0.0254 m horizontal pipe.
2. Volume of Fluid (VOF) multiphase model with RNG k-ε two equations turbulent model was selected among other different multiphase and turbulent models based on the convergence, prediction off the oil-water stratified flow pattern and the smoothness of the interface.
3. Mesh independent study has been achieved to decide on the optimum mesh size to be used in the simulation process.
4. Pressure prediction base on different flow velocity have been observed. It can be seen that as velocity increases, the pressure gradient also increases.
5. The pressure prediction will be extended to examine the effect from different water volume fraction.

ACKNOWLEDGEMENTS

The authors would like to convey a great appreciation to Faculty of Mechanical Engineering, Universiti Teknologi Malaysia and Ocean and Aerospace Engineering Research Institute, Indonesia.

REFERENCE

1. O.Shoham, Y.Taitel, “*Stratified turbulent-turbulent gas-liquid flow in horizontal and inclined pipes*”, AIChE J. 30 (2) (1984) 377-385.
2. R.I, Issa, “*Prediction of turbulent, stratified, two-phase flow in inclined pipes and channels*”, Int. J. Multiphase Flow 14 (1) (1988) 141-154.
3. C.H. Newton, M. Behnia, “*Numerical calculation of turbulent stratified gas-liquid pipe flows*”, Int. J. Multiphase Flow 24 (5) (1998) 141-154.
4. Hui Gao, Han-Yang Gu and Lie-Jin Guo, “*Numerical study of stratified oil-water two-phase turbulent flow in a horizontal tube*”, Int. J. heat and mass transfer, (46) (2003) 749-754
5. G. Elseth, H.K. Kvandal, M.C. Melaaen, “Measurement of velocity and phase fraction in stratified oil-water flow”, *International Symposium on Multiphase Flow and Transport Phenomena*, Antalya, Turkey, pp.206-210, 2000
6. Awal, Mohammad R., Zughbi, Habib D., Razzak, Shaikh A., Al-Majed, Abdulaziz A., and Al-Yousef, hasan Y., “*Liquids phase holdup and separation characteristics as a function of well inclination and flowrate*”, SPE SA, 2005, 14-16 May.
7. Carlos F. “*Modelling of oil-water flow in horizontal and near horizontal pipes*”, PhD Thesis, Tulsa University, 2006.
8. Al-Yaari, M., and Abu-Sharkh, “*CFD Prediction of stratified oil-water flow in a horizontal pipe*”, Asian Transactions on Engineering Volume 01 Issue 05, November 2011



(a) 46631 **(b) 79488**

(c) 104584 **(d) 142374**

Figure 3: Oil volume fraction contours at pipe length (z = 0.5 m)

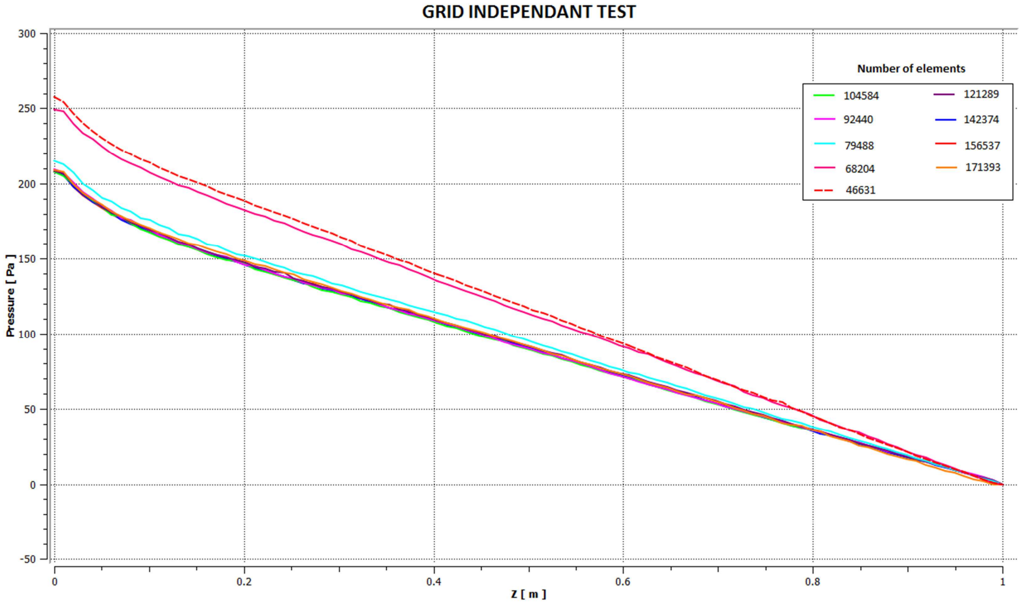


Figure 4: Optimum mesh size at unchanged pressure profile

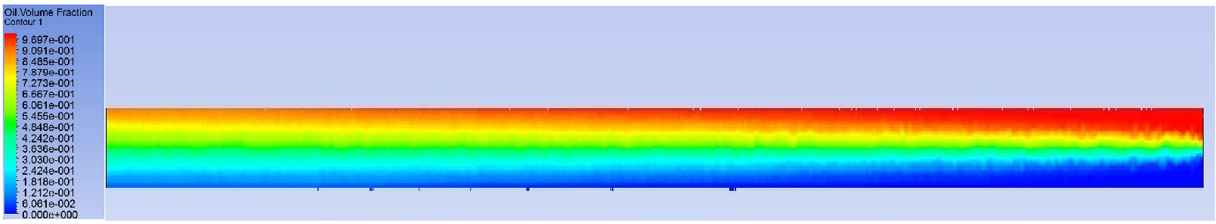


Figure 5: Stratified Oil-water flow simulation

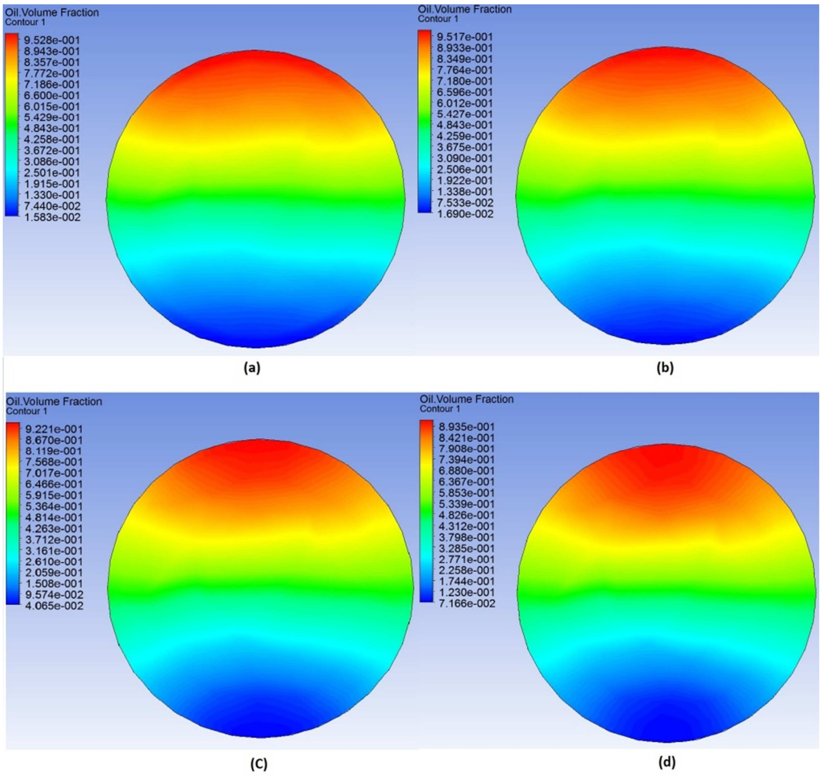


Figure 6: Oil volume fraction contours at pipe length (z = 0.5 m); (a) 1.1 m/s (b) 0.8 m/s (c) 0.5 m/s (d) 0.2 m/s

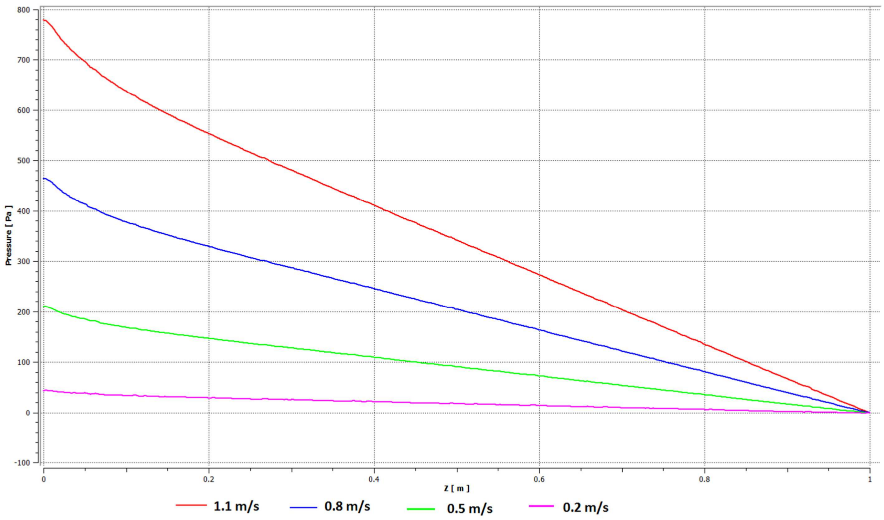


Figure 7: Pressure profile at each flow velocity

Journal of Subsea and Offshore
-Science and Engineering-, Vol.2

June 20, 2015

Buckling Criteria for Subsea Pipeline

Abdul Khair .J,[a,b,*]

[a)] *Department of Aeronautic, Automotive and Ocean Engineering, Mechanical Engineering, Universiti Teknologi Malaysia*
[b)] *Ocean and Aerospace Research Institute, Indonesia*

*Corresponding author: abdulkhairjunaidi@gmail.com

Paper History

Received: 20-May-2015
Received in revised form: 5-June-2015
Accepted: 19-June-2015

ABSTRACT

Oil and gas production in subsea operation continues to the extreme depth. Harsh environment and severe operation of oil and gas transportation due to high pressure and temperature become crucial for pipeline transportation. Consequently, The pipelines will deform to buckle shape which affect to integrity of pipeline. This phenomenon should be considered in design of pipeline to provide reliability of pipeline operation during time life period. The design result of pipelines is according to DNV F 101 whereas the magnitude of pipeline curvature will validate by ANSYS 14 to ensure pipeline reliability.

KEY WORDS: *Pipeline; Buckle; Expansion.*

NOMENCLATURE

HPHT	High Pressure and High Temperature
WISP	Wet Insulated Single Pipeline
FEA	Finite Element Analysis
LRFD	Load Resistance Factor Design

1.0 INTRODUCTION

Subsea production continues to the extreme depth of water. At this depth, the technical challenge of subsea system will be tight to comply with existing codes, moreover extreme pressure and temperature of crude oil is needed to transport from wet well to termination of loading. The pipelines are subjected to axial compressive forces which will cause the pipelines to expand, consequently the pipelines experience a deformation to buckle for certain size. Pipeline expansion should be allowed to accommodate the lateral movement of pipeline. Buckling is instability of pipeline structure that may be going to a failure if the curvature of buckling mode exceeds the pipeline strength.

2.0 OBJECTIVE OF DESIGN

The objective is to provide acceptance design for subsea pipeline which focus on buckling mode related to load response due to pressure and temperature. To be able to understand the buckling phenomena, an initial imperfection of pipeline at designated location along the line will be defined. The selection of material, pipe wall thickness and pressure containment corresponds to the limit state design of pipeline which refers to API RP 1111 and the load effect to the structure will comply with DNV OS F101.

3.0 LITERATURE REVIEW

Design of pipeline is required accurate test result for local buckling collapse subjected to bending loads which exceed the limit state of bending moment capacity. The minimum wall thickness is determined based on maximum allowable stress under design pressure. The design of pipeline is aimed to keep in safe during construction and operation and meet the life time period. The anomalous value of the axial tensile and compressive strain was obtained on the pipe test. Difference result derived from the test on pipe to the simple bending theory become design factor parameter to contribute to the understanding of crucial limit state for the design of onshore and offshore pipeline (F. Guarracino, 2007).

Subsea pipeline system operates under HPHT. Due to soil restraint, the pressure and thermal expansion can generate a significant level of compression that can cause global buckling in the pipeline. Global buckling is generally in lateral direction, although it can be started as an upheaval buckling. The two methods are applied to control the pipeline thermal expansion and lateral buckling by utilizing sleepers and buoyancy along the pipeline route. It uses two parallel positioned sleepers space in short distance. To further assess the pipeline buckling response and assist the selection of the thermal mitigation method, a series of numerical analysis were performed for a WISP through FEA. The FEA model length was set for 3,000 m. Buoyancy length and buoyancy force is analysed against the critical buckling. The presented study indicated that both sleeper and buoyancy section can be the viable solutions for thermal load mitigation. (Jason Sun,Pauljukes June 2012.)

4.0 DESIGN OF PIPELINE

The design of subsea pipeline must comply with the pipeline design codes such as ANSI/ASME B31.4, API RP 1111, DNV Design Guidelines. The pipeline standard gives the strict requirements for design, materials, construction, operation and maintenance to assure that the pipelines are safe to be operated during lifetime period without any failures or structure instabilities occurred such as buckling, fatigue, out of roundness and excessive free spans and etc. The DNV OS F101 gives the design requirement for pressure containment which is called LRFD. The LRFD principle refers to the design method in structural engineering that the actual load does not exceed the design resistance of the pipeline.

The DNV provides the formula to restrict compressive strain which does not exceed the design strain. The parameters are used include minimum internal pressure, external pressure and girth weld factor and choosing the value based on ratio D/T.

$$\varepsilon_{sd} \leq \ \varepsilon_{Rd} = \frac{\varepsilon_c(t_2, p_{min} - p_e)}{\gamma_\varepsilon} \qquad (1)$$

Where:
ε_{sd} = design compressive strain
$\varepsilon_c(t_2, p_{min} - p_e =$
$0.78\left(\frac{t}{D} - 0.01\right).\left(1 + 5.75\,.\frac{p_{min} - p_e}{p_b(t)}\right).\alpha_h^{-1.5}.\alpha_{gw}$
p_{min} = minimum internal pressure
p_e = external pressure, ρgh. (3)
$\alpha_\mathrm{h} = (\frac{\mathrm{R_{t\,0.5}}}{\mathrm{R_m}})_\mathrm{max}$ (4)
$(\frac{\mathrm{R_{t\,0.5}}}{\mathrm{R_m}})_\mathrm{max} = 0.93$

4.1 Pipeline Expansion

The amount of the pipeline expansion is an important design factor used in designing absorption devices such as loop or sleeper. The movement of pipeline expansion due to internal pressure and temperature are normally occurred in the pipeline, but the impacts of expansion movement will affect the pipe length at the end of pipeline. Forces result from internal load and temperature can be calculated as follow:

Force due to temperature change;

$$F_t = \alpha . E . A_s . \Delta T \qquad (2)$$

Force due to pressure change;

$$F_p = P . A_i \qquad (3)$$

Force due to Poisson contraction;

$$F_v = -v . A_s . \sigma_h \qquad (4)$$

Force due to soil friction resistance;

$$F_f = \int_0^x \mu . W_s . dx = \mu . W_s . L_a \qquad (5)$$

By equilibrium of the above forces for pipeline can be written:

$$F_t + F_p + F_v = F_f \qquad (6)$$

The anchor length can be obtained using the above equation

$$L_a = \frac{1}{\mu W_s}(\alpha E A_s \Delta T + P A_i - v A_s \sigma_h) \qquad (7)$$

The stress induced by the thermal and pressure expansion including the end cap effect can be written as:

$$\sigma_h = \frac{PD}{2t} \qquad (8)$$

$$\sigma_{L1} = \frac{P A_i}{A_s} - \frac{\mu . W_{S.} x}{A_S} \mathrm{if} x \ < L_a \qquad (9)$$

This condition is for unrestrained line that the stress limitation to maintain the expansion stress σ_e should not exceed 0.72% of the SMYS.

$$\sigma_e = (\sigma_L^2 + 4\sigma_t^2)^{0.5} \leq 0.72\sigma_y \qquad (10)$$

4.2 Configuration of Buckling

When this expansion is restraint by axial friction between the pipeline and the soil furthermore an axial force will develop to be lateral movement in the pipeline. Subsea pipelines could buckle upward or sideway direction. The direction of movement will depend on the pipe-soil resistance. The effective axial force in the pipeline is given by:

$$P_0 = (1 - 2v)\frac{\pi}{4} D^2 \Delta p + \ \pi D t E \alpha \Delta T \qquad (11)$$

The configuration of the buckle can be calculated by solving the following expression for buckle length L,

$$P_0 = P + k_2 \mu W L \left[\sqrt{1 + k_2 \frac{E A \mu_l^2 W L^5}{\mu (EI)^2}}\right] \qquad (12)$$

P= Compressive effective axial force within the buckle, given by

$$P = k_1 \frac{EI}{L^2} \qquad (13)$$

The maximum amplitude of the buckle can be determined

$$y = k_4 \frac{\mu_L W L^4}{EI} \qquad (14)$$

The maximum bending moment is calculated by

$$M = k_5 \mu_L W L^4 \quad (15)$$

Table 1 Buckling Constant

Mode	k_1	k_2	k_3	k_4	k_5
1	80.76	6.391×10^{-5}	0.5	2.407×10^{-3}	0.06938
2	$4\pi^2$	1.743×10^{-4}	1.0	5.532×10^{-3}	0.1088
3	34.06	1.668×10^{-4}	1.294	1.032×10^{-2}	0.1434
4	28.20	2.144×10^{-4}	1.608	1.047×10^{-2}	0.1483

4.3 Temperature Profile

The temperature along a subsea pipeline is conducted by heat flow in the pipeline. The temperature should be maintained at certain temperature to avoid wax deposit on the pipe wall. Three types of temperature profile along the pipeline as shown below:

1. Exponential Temperature Decay

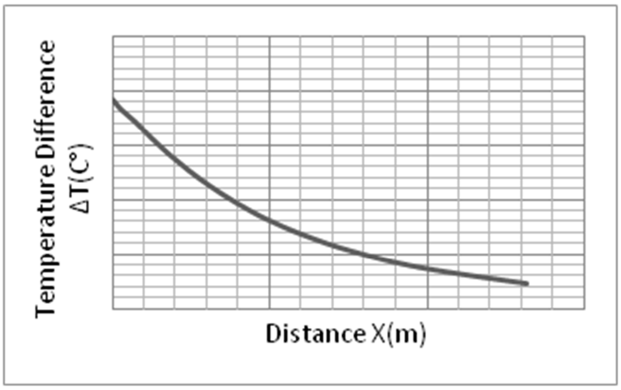


Figure 1: Exponential Temperature Decay.

2. Linear Temperature Decay

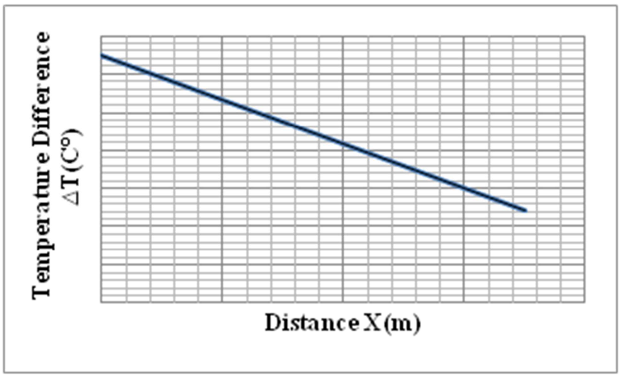


Figure 2: Linear Temperature Decay.

3. Uniform Temperature Decay

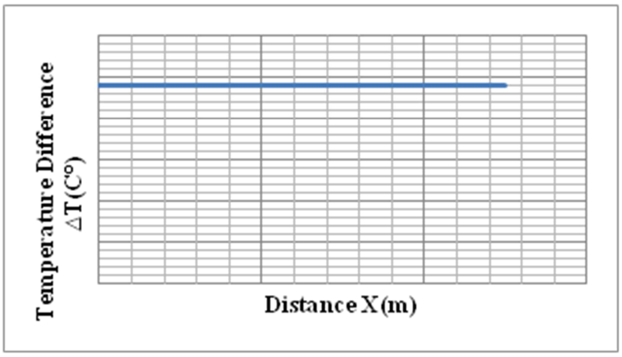


Figure 3: Uniform Temperature Decay.

5.0 ENGINEERING DATA

Input data comprise with Material properties of pipeline based on API 5L specification is shown at table 2 (Pipeline Properties), operating condition and environment condition. High strength steel in grades X80 is selected to assist companies to assess in reducing pipeline weight and demand more economical transportation lines. This simulation refers to the behavior of material properties of grade X80 which experience imperfection as consequent of high pressure and temperature operation.

Table 2: Pipeline Properties.

Parameter	Unit	Value
Outside Diameter	mm	762
Wall thickness	mm	20*
Pipe Material Grade	-	X80
Steel Density	Kg/m^3	7850
SMYS	MPa	551
SMTS	MPa	620
Poisson ratio (*v*)	-	0.3
Young's Modulus (E)	GPa	207
Thermal Expansion Coef.(α)	C^{-1}	1.17E-05
Internal Pressure	MPa	15
External Pressure	MPa	10
Internal Temperature	^{0}C	70
External Temperature	^{0}C	10

Based upon the calculation of internal loads, that provides a simple model of one way buckling, the model is described in detail about the magnitude of curvature and maximum displacement of pipeline as shown in figure 5.

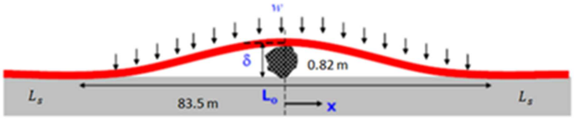


Figure 5: Buckle Curvature.

6.0 PIPELINE MESH IN ANSYS

ANSYS Meshing provides multiple mesh control to generate a mesh and to set an option on how the geometries are meshed. The meshing automatically sets default mesh size on pipeline geometry. A 3D Finite Element model and mesh was created to obtain proper solution in pipeline design. The pipeline length is 83.5 meter and it is not complicated structure to generate mesh element size resulting 163152 nodes and 23296 elements as shown in Figure.6.

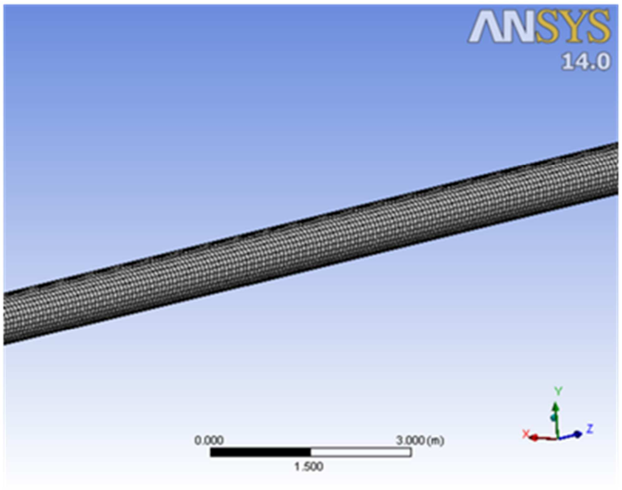


Figure 6: Pipeline Meshing

7.0 RESULT AND DISCUSSION

ANSYS simulation were applied to the pipeline model as the figure 7 shows pipeline deformation along with z-axis that subjected to axial compressive load. Large deformation indicates the elongations are occurred underneath the slope region whereas the upper side formed compressive deformation caused by buckle curvature upward.

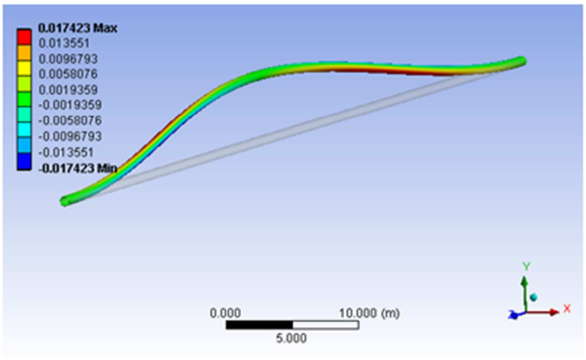


Figure 7: Deformation z-axis.

Pipeline expansion caused by pressure and temperature resulted vertical height of displacement at y-axis as shown in figure 8. The maximum height of buckle curvature is 0.89 meter, whereas the results of theoretical design that the height of curvature as shown in figure 5 (Buckle Curvature) is 082 meter.

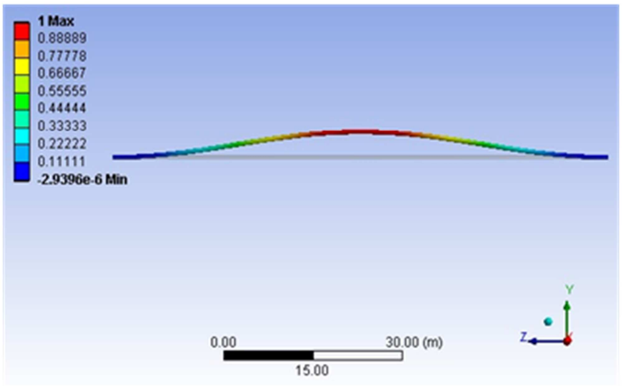


Figure 8: Displacement y-axis.

8.0 CONCLUSION

Pipelines experience elongation due to high internal pressure and temperature to transport the crude oil. The design allows the pipeline expand lateral or upheaval at designated location to relieve the pipeline expansion. The design result is appropriate with ANSYS Workbench to validate the modeling

REFERENCES

(1) F. Guarracino, M. Fraldi, A. Giordano.2007.*Analysis of testing methods of pipelines for limit state design*, Dipartimento di IngegneriaStrutturale, Università di Napoli ``Federico II", 80125 Napoli, Italy.

(2) Jason Sun,Pauljukes. 2012. *Thermal Expansion/Global Buckling Mitigation of HPHT Deepwater Pipelines Sleeper or Buoyancy.* International offshore and polar engineering Conference ,Greece June

(3) Jaswar, Abdul Khair.J, H.A.Rashid. 2013, *Expansion of Deep Water Subsea Pipeline*, Applied in The International Conference on Marine Safety and Environment.
(4) Jaswar, Abdul Khair.J, Hashif. 2013, .*Effect of Axial Force Concept in Offshore Pipeline Design*, The International Conference on Marine Safety and Environment.
(5) Palmer, A. C. and R. A. King. 2008, *Subsea Pipeline Engineering*, Pennwell Corporation.
(6) Palmer, D.kaye and Associates. 1966.*Lateral Buckling of Subsea Pipeline; Comparison Between Design and operation*, A Division of SAIC Limited 13 Bon Accord Square United Kingdom.
(7) James G.A. 1998.*A Simplified Analysis of Imperfect Thermally Buckled Subsea Pipelines*, International Journal of Offshore and Polar Engineering Vol. 8, No. 4, December 1998 (ISSN 1053-5381) Copyright © by The International Society of Offshore and Polar Engineers.
(8) Det Norske Veritas. DNV 2007. Offshore standard DNV-OS-F101: submarine pipeline systems. Hovik, Norway : Det Norske Veritas, DNV
(9) BAI, Y. and Q. BAI. 2010, *Subsea Engineering Handbook*. USA, ELSEVIER.
(10) Collberg, O. F. a. L. 2005. *Influence of Pressure in Pipeline Design – Effective Axial Force*. 24th International Conference on Offshore Mechanics and Arctic Engineering (OMAE 2005). .

Journal of Subsea and Offshore
-Science and Engineering-, Vol.2

June 20, 2015

Prediction of Motion Responses of Rounded-Shape FPSO using Diffraction Potential

C.L.Siow,[a] J. Koto,[a,b,*], M.Pauzi,[c] H.Yasukawa,[d] A.Matsuda,[e], D.Terada,[e]

[a] *Department of Aeronautics, Automotive and Ocean Engineering, Faculty of Mechanical Engineering, Universiti Teknologi Malaysia*
[b] *Ocean and Aerospace Research Institute, Indonesia*
[c] *Marine Technology Center, Universiti Teknologi Malaysia..*
[d] *Department of Transportation and Environmental Systems, Hiroshima University, Japan*
[e] *National Research Institute of Fisheries Engineering (NRIFE), Japan*

*Corresponding author: jaswar@mail.fkm.utm.my and jaswar.koto@gmail.com

Paper History

Received: 20-May-2015
Received in revised form: 5-June-2015
Accepted: 19-June-2015

ABSTRACT

This paper reviewed the capability of the proposed programming coded based on diffraction potential theory to predict a floating structure's motion response. The proposed programming code was applied to prediction motion responses of rounded-shape FPSO in surging, heaving and pitching directions. This paper briefly presents the procedure to apply the diffraction potential theory to simulate the rounded-shape FPSO motion responses. Results of simulation were compared with ANSYS AQWA software as bench mark. It found that the simulation results by the proposed programming code agree with the ANSYS one.

KEY WORDS: *Wave Response, Diffraction Potential, ANSYS; KVLCC Ship.*

NOMENCLATURE

$\Phi(x, y, z)$	Velocity Potential in x, y, z directions
$G(P; Q)$	Green Function
F_D	Drag Force
R	Horizontal Distance
K	Wave Number

1.0 INTRODUCTION

Behavior or a floating structure using Round Shape FPSO was studied by Lamport and Josefsson in year 2008. They were carried a research to study the advantage of round shape FPSO over the traditional ship-shape FPSO [1]. The comparisons were made to compare motion response, mooring system design, constructability and fabrication, operability, safety and costing between both the structures. One of the finding on their study is the motions of their designed structures are similar at any direction of incident wave with little yaw excitation due to mooring and riser asymmetry. Next, Arslan, Pettersen, and Andersson (2011) are also performed a study on fluid flow around the round shape FPSO in side-by-side offloading condition. FLUENT software was used to simulate three dimensional (3D) unsteady cross flow pass a pair of ship sections in close proximity and the behavior of the vortex-shedding around the two bluff bodies [2]. Besides, simulation of fluid flow Characteristic around Rounded-Shape FPSO by self-develop programming code based on RANs method also conducted by A. Efi et al.[3].

As presented by Siow.et al. [6], their finding found that the diffraction potential theory is less accurate to predict the floating structure heave motion response when the wave frequency is close to the structure's natural frequency. In this situation, the heave response calculated by the diffraction potential theory is significantly higher compared to experimental result due to the low damping represented by the theory [9].

In order to improve the heave motion predict by the diffraction potential theory, Siow. et al. tried to increase the damping coefficient by adding viscous damping into the motion equation.

In his study, the viscous damping is treated as an extra matrix and can be added into the motion equation separately [6]. Besides this, Siow et al. also tried to integrate the linearized Morison drag equation with diffraction potential theory. The linear Morison drag equation would modify both the damping term and exciting force in the motion equation compared to the viscous damping correction method which only modified the damping term in motion equation. The accuracy of the modification solutions are also checked with the semi-submersible experiment result which was carried out at the towing tank of the Universiti Teknologi Malaysia [10]. The 6-DOF Round Shape FPSO motion result calculated by this method and the comparison of result between the proposed methods with experiment result was published by Siow et.al in year 2015 [11].

This paper is targeted to review the accuracy of diffraction potential theory in order to evaluate the motion response of a ship. The diffraction potential theory estimates wave exciting forces on the floating body based on the frequency domain and this method can be considered as an efficient one to study the motion of large size floating structure with acceptable accuracy. The accuracy of the diffraction potential method to predict the structures response was also detailed studied. The good accuracy of this diffraction theory applied to large structures is due to the significant diffraction effect that exists in the large size structure in wave [4]. In this study, the motion response of KVLCC2 ship is simulated by the diffraction potential theory and compared with ANSYS as bench marking.

2.0 NUMERICAL CALCULATION

2.1 Diffraction Potential

In this study, the diffraction potential method was used to obtain the wave force act on the Round Shape FPSO also the added mass and damping for all six directions of motions. The regular wave acting on floating bodies can be described by velocity potential. The velocity potential normally written in respective to the flow direction and time as below:

$$\Phi(x,y,z) = Re\left[\phi(x,y,z)e^{iwt}\right] \qquad (1)$$

$$\phi(x,y,z) = \frac{g\varsigma_a}{iw}\{\phi_0(x,y,z)+\phi_7(x,y,z)\} + \sum_{j=1}^{6} iwX_j\phi_j(x,y,z) \qquad (2)$$

where,

g : Gravity acceleration
ς_a : Incident wave amplitude
X_j : Motions amplitude
ϕ_0 : Incident wave potential
ϕ_7 : Scattering wave potential
ϕ_j : Radiation wave potential due to motions
j : Direction of motion

From the above equation, it is shown that total wave potential in the system is contributed by the potential of the incident wave, scattering wave and radiation wave. In addition, the phase and amplitude of both the incident wave and scattering wave are assumed to be the same. However, radiation wave potentials are affected by each type of motions of each single floating body in the system, where the total radiation wave potential from the single body is the summation of the radiation wave generates by each type of body motions such as surge, sway,heave,roll, pitch and yaw,

Also, the wave potential $\emptyset$ must be satisfied with boundary conditions as below:

$$\nabla^2\emptyset = 0 \quad for\ 0 \le z \le h \qquad (3)$$

$$\frac{\partial\emptyset}{\partial z} + k\emptyset \quad at\ z = 0 \ \ (k = \frac{w^2}{g}) \qquad (4)$$

$$\frac{\partial\emptyset}{\partial z} = 0 \quad at\ z = h \qquad (5)$$

$$\emptyset \sim \frac{1}{\sqrt{r}}e^{-ik_0 r} \ \ should\ be\ 0\ if\ r\ \infty \qquad (6)$$

$$\frac{\partial\phi_7}{\partial n} = -\frac{\partial\emptyset_0}{\partial n}\ on\ the\ body\ boundary \qquad (7)$$

2.2 Wave Potential

By considering the wave potential only affected by model surface, S_H, the wave potential at any point can be presented by the following equation:

$$\emptyset(P) = \iint_{S_H}\left\{\frac{\partial\emptyset(Q)}{\partial n_Q}G(P;Q) - \emptyset(Q)\frac{\partial G(P;Q)}{\partial n_Q}\right\}dS(Q) \qquad (8)$$

where*P* =*(x, y, z)* represents fluid flow pointed at any coordinate and $Q = (\xi,\eta,\varsigma)$ represent any coordinate, *(x, y, z)* on model surface, S_H. The green function can be applied here to estimate the strength of the wave flow potential. The green function in eq. (8) can be summarized as follow:

$$G(P;Q) = -\frac{1}{4\pi\sqrt{(x-\xi)^2+(y-\eta)^2+(z-\zeta)^2}} + H(x-\xi, y-\eta, z+\zeta) \qquad (9)$$

where $H(x-\xi, y-\eta, z+\zeta)$ in eq. (9) represent the effect of free surface and can be solved by second kind of Bessel function.

2.3 Wave Force, Added Mass and Damping

The wave force or moment act on the model to cause the motions of structure can be obtained by integral the diffraction wave potential along the structure surface.

$$E_i = -\iint_{S_H}\phi_D(x,y,z)n_i dS \qquad (10)$$

where, ϕ_D is diffraction potential, $\phi_D = \phi_o + \phi_7$

Also, the added mass, A_{ij} and damping, B_{ij} for each motion can be obtained by integral the radiation wave due to each motion along the structure surface.

$$A_{ij} = -\rho\iint_{S_H} Re\left[\phi_j(x,y,z)\right]n_i dS \qquad (11)$$

$$B_{ij} = -\rho w\iint_{S_H} Im\left[\phi_j(x,y,z)\right]n_i dS \qquad (12)$$

$n_i$in eq. (10) to eq. (12) is the normal vector for each direction of motion, i = 1~ 6 represent the direction of motion and j = 1~6 represent the six type of motions. The motion equation is shown as follows:

$$(m+m_a)\ddot{X}_z + (b_p)\dot{X}_z + kx = F_p \tag{13}$$

3.0 SIMULATION RESULTS OF ROUNDED-SHAPE FPSO

The objective of this paper is reviewing motion responses of Rounded-Shape FPSO estimated by the diffraction potential theory. The designed Round Shape FPSO model has the diameter at the draft equal to 1.018 meters and draught of 0.2901 meters. The model was constructed from wood following the scale of 1:110. Upon the model complete constructed, inclining test, and roll decay test were conducted to identify the hydrostatic particular of the dimension and measured data of the model was summarized as in Table 2 and Figure.1.

Table 1: Particular of Round Shape FPSO

Symbol	Model
Diameter (m)	1.018
Depth (m)	0.4401
Draught(m)	0.2901
Free board(m)	0.150
Displacement (m^3)	0.2361
Water Plan Area (m^2)	0.8139
KG (m	0.2992
GM (m)	0.069

Figure 1: Model of Rounded-Shape FPSO

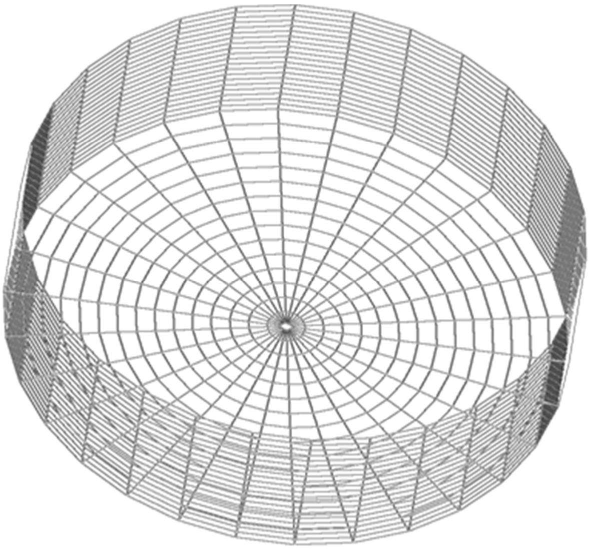


Figure 2: Meshing for Round Shape FPSO model

The motion responses of Rounded-Shape FPSO calculated by the diffraction potential theory and ANSYS Diffraction method are presented in Figures 3 ~ 5. From the figures, it can be seen that tendency by the diffraction potential theory the diffraction potential theory return the same result as the result predicted by ANSYS software. The observation also proved that the self-developed diffraction potential coding is developed based on the diffraction potential theory correctly.

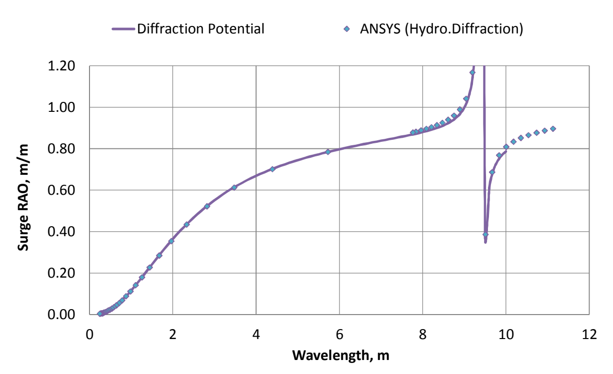


Figure 3: Surge motion response of Rounded-Shape FPSO predicted by diffraction potential theory and ANSYS AQWA software.

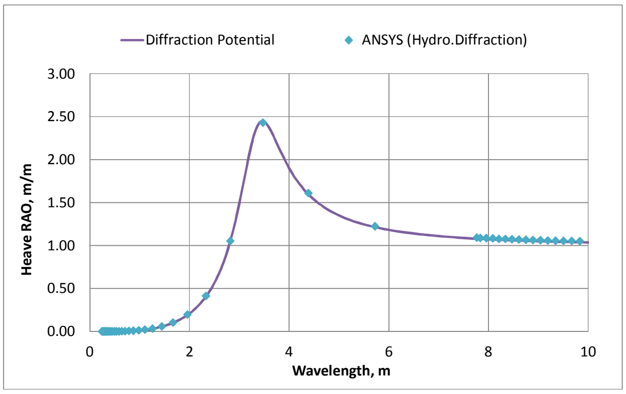


Figure 4: Heave motion response of Rounded-Shape FPSO predicted by diffraction potential theory and ANSYS AQWA software.

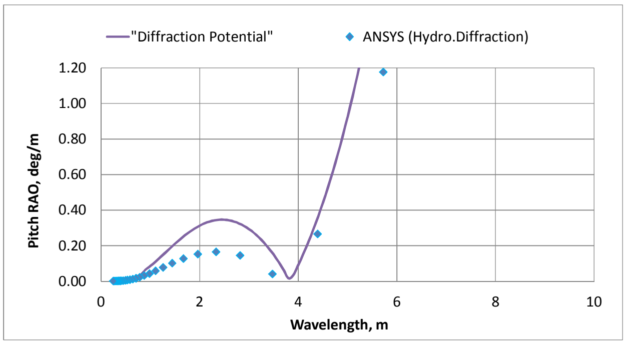


Figure 5: Pitch motion response of Rounded-Shape FPSO predicted by diffraction potential theory and ANSYS AQWA software.

5.0 CONCLUSION

In conclusion, this paper reviewed the tendency of heave motion response predicted by the proposed diffraction potential theory with Morison drag term correction method. In the beginning, the FPSO heave motion response predicted by the self-developed programming was compared to the predicted result by ANSYS AQWA. The comparison showed that the self-developed diffraction potential coding have the same performance as ANSYS AQWA software where both method provided same tendency of result and almost similar response amplitude at any wavelength.

ACKNOWLEGMENT

The authors are very grateful to Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, Ocean and Aerospace Research Institute, Indonesia, Department of Transportation and Environmental Systems, Hiroshima University, Japan and National Research Institute of Fisheries Engineering (NRIFE), Japan for supporting this research.

REFERENCE

1. Lamport, W. B. and Josefsson, P.M.(2008). The Next Generation Of Round Fit-For-Purpose Hull Form FPSOS Offers Advantages Over Traditional Ship-Shaped Hull Forms,*2008 Deep Gulf Conference*, December 9-11, New Orleans, Louisiana, USA.
2. Arslan, T., Pettersen, B. and Andersson, H.I. (2011). *Calculation of the Flow Around Two Interacting Ships*, Computational Methods in Marine Engineering IV, L.Eça, E. Oñate, J. García, T. Kvamsdal and P. Bergan (Eds.), pp. 254-265.
3. Afrizal, E., Mufti, F.M., Siow, C.L. &Jaswar. (2013). Study of Fluid Flow Characteristic around Rounded-Shape FPSO Using RANS Method. *The 8th International Conference on Numerical Analysis in Engineering*: 46 – 56. Pekanbaru, Indonesia.
4. Kvittem, M.I., Bachynski, E.E. & Moan, T. (2012). Effect of Hydrodynamic Modelling in Fully Coupled Simulations of a Semi-Submersible Wind Turbine. *Energy Procedia* 24.
5. Koto, J., Siow, C.L., Khairuddin, Afrizal, N.M., Abyn, H., Soares, C.G., (2014). Comparison of floating structures motion prediction between diffraction, diffraction-viscous and diffraction-Morison methods. *The 2nd International Conference on Maritime Technology and Engineering*. Lisboa, Portugal.
6. C.L Sow, Koto, J, Hassan Abyn, (2014), *Semi-Submersible Heave Response Study Using Diffraction Potential Theory with Viscous Damping Correction*, Journal of Ocean, Mechanical and Aerospace Science and Engineering, Vol. 5
7. Siow, C.L., Koto, J, Yasukawa, H., Matsuda, A., Terada, D., Soares, C.G., Zameri, M. (2014). Experiment Study on Hydrodynamics Characteristic of Rounded- Shape FPSO. *The 1st Conference on Ocean, Mechanical and Aerospace-Science and Engineering-*.Pekanbaru, Indonesia.
8. Siow, C. L., Koto, J, and Khairuddin, N.M. (2014). *Study on Model Scale Rounded-Shape FPSO's Mooring Lines*, Journal of Ocean, Mechanical and Aerospace Science and Engineering, Vol. 12.
9. Siow, C.L., Abby, H. &Jaswar (2013). Semi-Submersible's Response Prediction by Diffraction Potential Method.*The International Conference on Marine Safety and Environment*.Johor, Malaysia.
10. Siow, C.L., Jaswar, Afrizal, E., Abyn, H., Maimun, A, Pauzi, M. (2013). Comparative of Hydrodynamic Effect between Double Bodies to Single Body in Tank.*The 8th International Conference on Numerical Analysis in Engineering*.Pekanbaru, Indonesia.
11. Siow, C. L.,Koto, J., Yasukawa, H.,Matsuda,A. Terada, D., Guedes Soares, C., Muhamad Zameri bin Mat Samad and Priyanto,A.(2015). *Wave Induce Motion of Round Shaped FPSO*, Journal of Subsea and Offshore Science and Engineering, Vol. 1.
12. Cengel, Y. A, Cimbala, J. M. (2010). Fluid Mechanics Fundamentals and Application. 2nd Ed.

Published

ISOMAse
Resty Menara Hotel
Jalan Sisingamangaraja No.89
Pekanbaru-Riau, INDONESIA
http://www.isomase.org/

Edited

Building P-23, Room: 314
Department of Aeronautics,
Automotive & Ocean Engineering,
Faculty of Mechanical, Universiti
Teknologi Malaysia, MALAYSIA
http://web1.fkm.utm.my/

ISSN: 2442-6415

9 772442 641011

Printed

Teknik Mesin
Fakultas Teknik,
Universitas Riau, INDONESIA
http://www.unri.ac.id/en

Organized

Ocean & Aerospace Research
Institute, Indonesia
Pekanbaru-Riau, INDONESIA
http://isomase.org/OCARI.php/

Supported

Mechanical Chapter of the
Institution of Engineers,
INDONESIA

Malaysian Joint Branch Royal
Institution of Naval Architects &
Institute of Marine Engineering,
Science and Technology
-Southern Chapter (MJB RINA
&IMarEST – SC)-