Fluid Structure Interaction (FSI): Study of Multiphase Flow at T-Junction

Siti Mayamin Binti Mohamad Zaini, Assc. Prof. Ir. Dr. Nadiahnor Binti Md Yusop

Faculty of Chemical Engineering, Universiti Teknologi MARA

***Abstract*— In oil and gas industry, multiphase flow is a common phenomenon of the flow happened during the production phase. Multiphase flow is considered as a complex flow regime and need extra analysis tool to separate the fluid. The example of multiphase flow are gas-liquid flow, liquid-solid flow and solid-gas flow. Hence, pipeline is the most important equipment in industrial application to provide a good channel for the transportation of the multiphase flow. T-junction is the one of the important component of the pipeline system. The simulation of the multiphase flow when entering the T-junction is analyse by using Computational Fluid Dynamic (CFD). Eulerian model is utilized using software ANSYS FLUENT Workbench 15.0. The paper presents the modelling of the multiphase flow and the deformation of the structure when the fluid is impacting the T-junction. The Fluid Structure Interaction (FSI) of the multiphase flow on the T-junction that causing to the erosion is investigated.**

***Keywords*— Fluid Structure Interaction (FSI), Computational Fluid Dynamic (CFD), Multiphase flow, T-junction, Phase Redistribution**

I. Introduction

Multiphase flow can be defined as more than one flow occurs in a flow field such as solid, liquid or gas stream. Multiphase flow is a kind of simultaneous flow of the materials with different states of phases as mentioned above. It also identified as the materials with different chemical properties but still in the same states or phase such as oil droplets in water. Multiphase flow can be classified as either two phase flow (i.e. solid-liquid flow, solid-gas flow, liquid-liquid flow and liquid gas flow) or three phase flow (i.e. gas-liquid-solid flow, gas-liquid-liquid flow and solid-liquid-liquid flow). The study of multiphase flow is very essential in energy related industry especially in oil and gas industry (Thermopedia.com, 2017). The fluid system is divided into two phases which are primary phase and secondary phase. Primary phase is a continuous phase and secondary phase is a material that is considered to be dispersed within continuous phase (Athulya & Miji Cherian, 2016).

In the pipeline system, T-junction are one of the important part (Athulya & Miji Cherian, 2016). Junctions between pipes have ability to split or mix the fluid when flowing to the junction (Thermopedia.com, 2017). When two phase flow enter at T-junction, both will separate. The heavier phase will flow towards the main stream and the lighter phase will flow into the side stream. Thus, the side stream will carry greater proportion of gas while the main stream will carry greater proportion of liquid stream (Athulya & Miji Cherian, 2016). T-junction is a very common pipe component network that is used widely to diverge the flow of fluids from the main flow pipe to the several side flow pipe. The also used to assemble the flow of the fluids from several side flow pipe to the main flow pipe. The pattern of the fluid flow may change at the T-junction depending on its inflow and outflow of the fluids (Vasava, 2007).

The Fluid Structure Interaction (FSI) will be those connection for a portion versatile or deformable structure for an internal or encompassing fluid stream (Siva & Sankar, 2015). FSI occurs when the fluid is flowing causes the deformation. Three forces that can lead deformation to happen are fluid flow encounters the structure, stresses and strains exerted on the solid object (Comsol.com, 2017). The pressure and the velocity of the flow and the material properties of the actual structure will effects on the deformation of the fluids either the deformation happened is large or small. The deformation or splitting of the fluids will change their boundary condition of the fluid flows. Multiphase models is used to demonstrate the physics concept of multiphase flow by using Computational Fluid Dynamics (CFD). This paper used Eulerian Model in ANSYS FLUENT Workbench 15.0 software to simulate the flow and phase distribution at the T-junction, thus completing the study of multiphase flow impacting at T-junction.

The redistribution occurs when the multiphase flow enters the T-junction of the pipe. This will cause the interaction mechanism in the T-junction, namely the sheer stress between the fluid and the pipe wall, axial movement of the pipe induced by the radial deformation due to pressure different and the pipe movement due to imbalance coupled forces at T-junction and boundaries (Athulya & Miji Cherian, 2016). The key parameters that affect the fluid redistribution are on the volume fraction, density, pressure difference and the superficial velocity of the liquid and gas flow in the flow pipe. The existence of more than one phase at the T-junction often contribute to the erosion and internal corrosion at T-junction of the flow pipe. The value of gas superficial velocity and liquid superficial velocity and the effect on the bend angle will give big impact on the rate of erosion.

II. METHODOLOGY

A. Computational Method

1) Physical Schematic Diagram

Physical schematic diagram will shows the geometry area of pipeline T-junction. In this paper, a study of T-junction on horizontal main pipeline with vertical T-junction of 90^0 is presented. The length of horizontal pipeline is 600mm while the length of vertical pipeline is 300mm. Outer and inner diameter are 50mm and 40mm respectively for both horizontal and vertical pipeline.

2) Grid Independences Analysis

In the numerical modelling, in order to get the best valid conclusion of the model performance, a grid independence analysis is carried out to minimize the numerical errors. This is describe in Best Practice Guideline (BPG) for validation of the CFD code. In general, the grid independence study is describes as the error of the solution within that particular range which it can be accepted at the end of the study (Liu et al., 2017). This study has been investigated to find the optimum number of element with four different no of element which are 10 000, 13 684, 15 874 and 16 875. These different number of element have features and same structural of mesh. The analysis were carried out between number of element and superficial velocity of the fluid flow. The result of grid independence analysis as Figure 1.

As the calculation result obtain from grid independence analysis, the value of velocity is increasing as the number of element increase. At one point, the velocity remains constant with increasing number of element. This prove that the graph has achieved the optimum number of element. Hence, 15 874 is chosen as the optimum element number for the calculation.

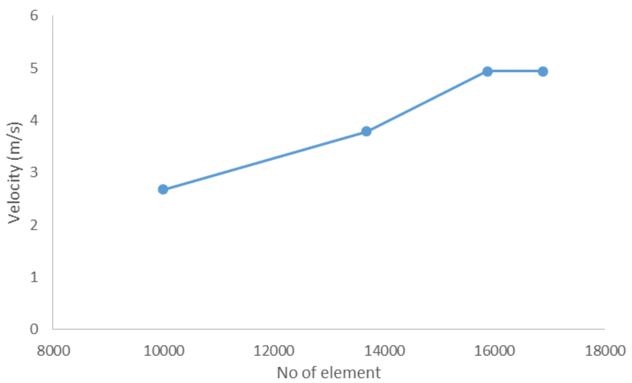


Figure 1: Grid Independences Analysis

B. *Numerical Simulation Method*

1) *Continuity Equation*

This continuity principles is derived from mass which are always conserved in fluid flow regime no matter of pipeline complexity or the fluid flow direction.

$$\frac{\partial p}{\partial t} + \nabla.(\rho U) = 0 \quad (3.1)$$

In the above continuity equation (Liu et al., 2017), ρ, U, t denotes as density fluid, velocity vector of the fluid and time respectively.

2) *Momentum Equation*

As stated in the Newton's second law, the rate of change of the body is equal to the net force acting on the body (Granger, 2012). The equation of conservation of momentum as follows (Rzehak & Kriebitzsch, 2015)

$$\frac{\partial}{\partial t}(\alpha_G \rho_G U_G) + \nabla.(\alpha_G \rho_G U_G.U_G) = -\alpha_G \nabla p + \nabla.(\alpha_G T_G) + \alpha_G \rho_G g + F_G \quad (3.2)$$

$$\frac{\partial}{\partial t}(\alpha_L \rho_L U_L) + \nabla.(\alpha_L \rho_L U_L.U_L) = -\alpha_L \nabla p + \nabla.(\alpha_L T_L) + \alpha_L \rho_L g + F_L \quad (3.3)$$

In the equation above, α, ρ, g and u are volume fraction of gas and liquid, density, gravitational force and superficial velocity of gas and liquid respectively.

3) *Mass Equation*

The mass conservation equation for both gas and liquid are as follow (Rzehak & Kriebitzsch, 2015). In this equation α, ρ, and u are refer to volume fraction of gas and liquid, density, gravitational force and superficial velocity of gas and liquid respectively.

$$\frac{\partial}{\partial t}(\alpha_G \rho_G) + \nabla.(\alpha_G \rho_G U_G) = 0 \quad (3.4)$$

$$\frac{\partial}{\partial t}(\alpha_L \rho_L) + \nabla.(\alpha_L \rho_L U_L) = 0 \quad (3.5)$$

4) *Energy Equation*

$$\dot{Q} + \sum_{in} \dot{m}_{in}(e_{in} + p_{in} v_{in}) = \frac{dMe}{dt} + \sum_{out} \dot{m}_{out}(e_{out} + p_{out} v_{out}) \quad (3.6)$$

Where $e = u + \frac{v^2}{2} + gH$

5) *k-ε Model*

As the flow the in pipe T-junction is consider as turbulent flow, the k-ε are adopted in the equation below (Chen et al., 2015).

$$\frac{\partial}{\partial t}(\alpha_f \rho_f k) + \nabla.(\alpha_f \rho_f u_f k) = \nabla.(\alpha_f \tau_k \nabla k) + \alpha_f G - \alpha_f \rho_f \varepsilon \quad (3.7)$$

$$\frac{\partial}{\partial t}(\alpha_f \rho_f \varepsilon) + \nabla.(\alpha_f \rho_f u_f \varepsilon) = \nabla.(\alpha_f \tau_\varepsilon \nabla \varepsilon) + \alpha_f \frac{\varepsilon}{k}(c_1 G - c_2 \rho_f \varepsilon) \quad (3.8)$$

6) *Multiphase Equation*

The volume flux or often call as superficial velocity U_i is defined as (Thermopedia.com, 2017).

$$U_i = \frac{V_i}{S} \quad (3.9)$$

Where S is refer to cross sectional area(m^2), V_i is refer to volume of the flow rate of that particular phase$\left(m^3/_s\right)$. The total superficial velocity U_t is given by (Thermopedia.com, 2017)

$$U_t = \sum_{i-1}^{n} U_i \quad (3.10)$$

Where n and U_i are refer to the number of phase present in the fluid and superficial velocity respectively. The author also state the average phase velocity as (Thermopedia.com, 2017)

$$u_i = \frac{U_i}{\varepsilon_i} = \frac{V_i}{S\varepsilon_i} \quad (3.11)$$

The flow quality x_i of the ith are as follows (Thermopedia.com, 2017)

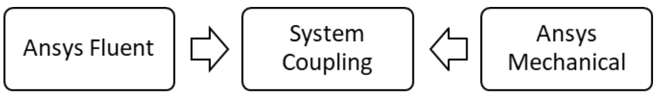
$$x_i = \frac{\dot{m}_i}{\sum_{i-1}^{n} \dot{m}_i} \quad (3.12)$$

Where the mass flux of the ith phase is denoted as$\dot{m}_i$. The density of the multiphase flow ρ_{MP} is mean mass of multiphase mixture per unit channel volume (Thermopedia.com, 2017).

$$\rho_{MP} = \sum_{i-1}^{n} \varepsilon_i \rho_i \quad (3.13)$$

C. *Simulation*

Simulation were performed on the geometry created as mentioned previously by applying boundary condition. This paper were conducted two simulation which are fluid domain and fluid structure interaction. These simulation were performed by using ANSYS Fluent Workbench 15.0 for fluid domain and ANSYS Mechanical Workbench 15.0 of Static Structural for fluid structure interaction.



1) *ANSYS Fluent Workbench 15.0*

This study illustrated the simulation using ANSYS Fluent fluid flow system in ANSYS Workbench to set up and solve multiphase turbulence fluid flow at T-junction. As the Eulerian Multiphase model was determined for this study because it is expected to make a more realistic prediction in this case, the computational effort required to solve multiphase flow problem. The number of transport equation being solved and the degree of coupling plays an important role for the required computational effort. The computational expenses will be high since Eulerian multiphase model has a large number of highly coupled transport equations.

Simulation was performed by applying the boundary condition. Water and air velocity were set as 1.53m/s and 1.6m/s respectively. Air volume fraction is 0.52, diameter of air bubble is 1 mm. The pressure is set as 0.5kPa. Flow rate weighting factor for both outlet_1 and outlet_2 are 0.2 and 0.8 respectively.

Realizable k-epsilon was chosen as the viscous model. For the solution parameter, Phase Coupled SIMPLE was taken as the solution scheme. Under Gradient of Spatial discretization, Least Square Cell Based was chosen. First order upwind scheme was selected to use for discretization for momentum, volume fraction, turbulent kinetic energy and turbulent dissipate rate.

2) ANSYS Mechanical Workbench 15.0

The study was illustrated the simulation using ANSYS Static Structural in ANSYS Workbench 15.0 to set up and solve the deformation of pipeline T-junction when loads is considered. The geometry and the set up for ANSYS Static Structural was drag from ANSYS Fluent. The auto-generated meshing was chosen.

III. RESULTS AND DISCUSSIONS

A. Phenomenon of Fluid Structure Interaction (FSI)

Result of numerical simulation of pipeline T-junction are represented. Working medium at given flow line as Figure 2 flows along OX axis which represent the horizontal main arm, separates and part of it enters along OY axis which represent the vertical downward branch arm. According to Figure 2, zones of maximal motion intensification appear in points of flow separation from the main arm at the center of T-junction and on neck of branch arm in place of separated flow strike.

The phenomenon of Fluid Structure Interaction (FSI) around the center of main arm OX axis T-junction and branch arm's neck OY axis were affected by the flow velocity, radial distribution of total pressure and turbulent kinetic energy. These will give effects on how the fluid structure redistribution around the constriction area. The continuity of these main effects is the result of the erosion rate in the internal pipe wall of pipeline T-junction.

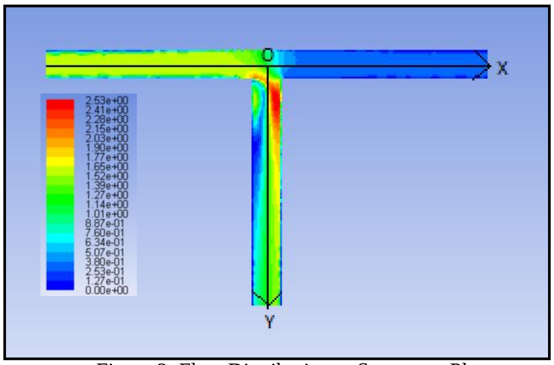


Figure 2: Flow Distribution at Symmetry Plane

1) The effect of flow velocity

According to Figure 3, deceleration of flow velocity zone was appeared at the opposite wall of branch arm, while acceleration of flow velocity zone was appeared at the branch arm's neck flow, separates from the main arm OX axis and strikes to the branch arm OY axis. By analyzing velocity profile on main arm OX line, it shows the disruption of flow axial symmetry near the flow of separation zone, with further reconstruction to axial symmetry at the T-junction outlet.

In flow redistribution area of axial symmetry disruption was appeared as it has smooth flow decreasing rate change. At the same time, at the area which close to the branch arm, axial velocity growth is observed with following sharp decrease to extreme values when the flow is crossing the branch arm OY axis. Radial velocity redistribution in T-junction is also a serious interest. It is sufficiently asymmetry across branch arm OY axis and reached maximal values in the zone of separated flow strikes on branch wall.

The Figure 4, 5 and 6 shows the contour distribution of fluid velocity at low, medium and high flow inlet respectively.

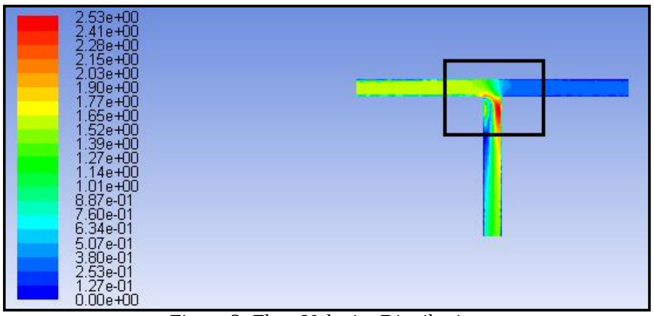


Figure 3: Flow Velocity Distribution

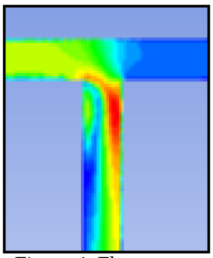


Figure 4: Flow Velocity Distribution at Low Velocity

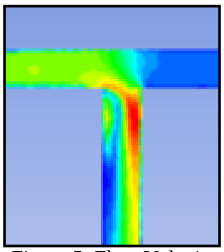


Figure 5: Flow Velocity Distribution at Medium Velocity

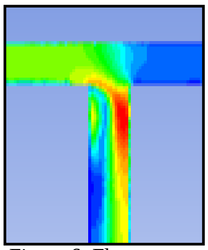


Figure 6: Flow Velocity Distribution at High Velocity

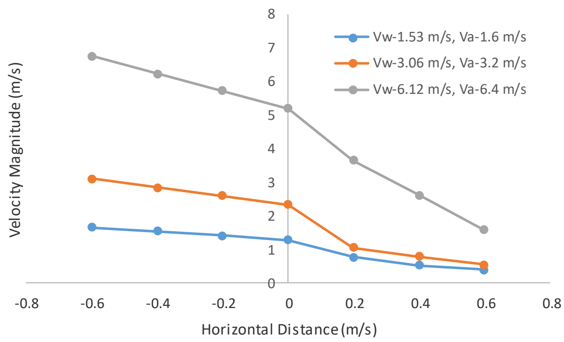


Figure 7: Various Flow Velocity Inlet across Main Arm

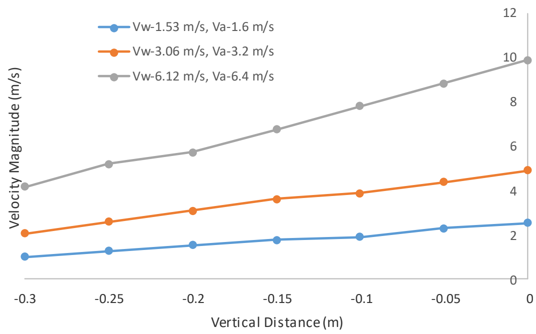


Figure 8: Various Flow Velocity Inlet across Branch Arm

2) Radial Distribution of Total Pressure

Variation of the pressure of fluid domain along the T-junction can be describe by the contour variation as shown Figure 9. As we can see from the contour below, the pressure distribution is continuously decreasingly along the main arm OX axis. The maximum pressure is at the early center of main arm OX axis and branch arm' neck OY axis, where the stagnation point is located.

This is due to the zero velocity of fluid flow that hit the internal wall of the pipe and causes high pressure produce at point (T) of interest compare to other parts in the pipeline.

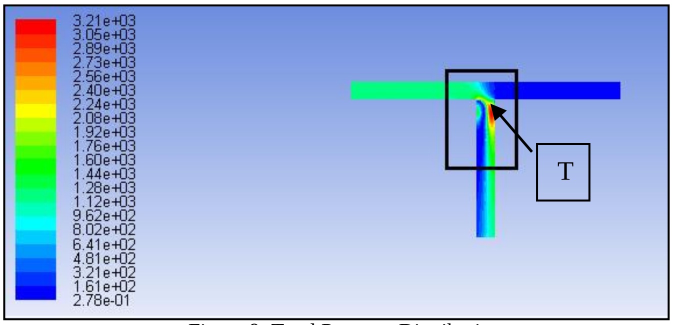


Figure 9: Total Pressure Distribution

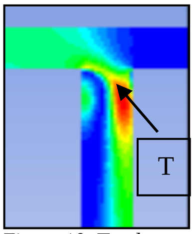


Figure 10: Total Pressure Distribution at Low Pressure

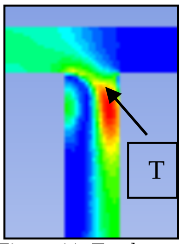


Figure 11: Total Pressure Distribution at Medium Pressure

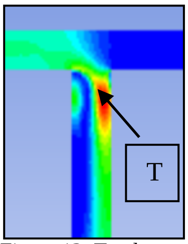


Figure 12: Total Pressure Distribution at High Pressure

The radial distribution of total pressure among three different pressure inlet can be explained through the plotted graph at Figure 13 and Figure 14. As the higher pressure inlet exerted to the pipe flow, the greater the radial pressure distributed along the pipeline T-junction. The same pattern of decreasing the pressure along the pipeline whether is it in main arm OX axis and branch arm OY axis between the difference pressure inlets of the mixture can be shown in Figure 13 and 14.

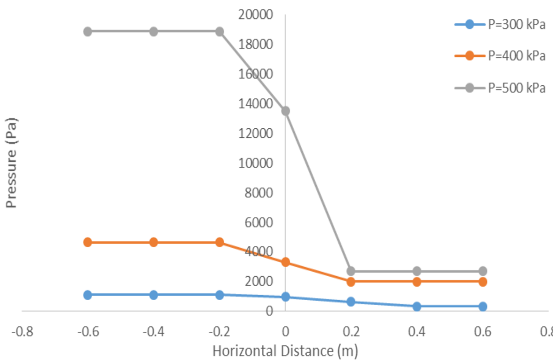


Figure 13: Radial Distribution of Total Pressure across Main Arm

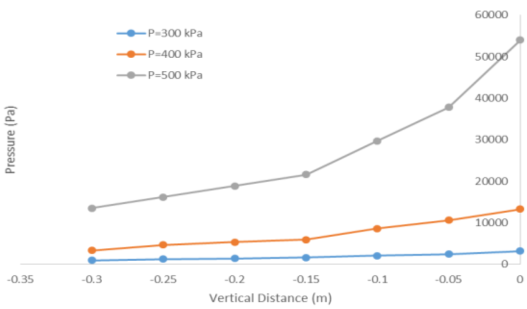


Figure 14: Radial Distribution of Total Pressure across Branch Arm

3) Effect on turbulent kinetic energy

According to turbulent kinetic energy in Figure 15, additional hydraulic resistance to fluid flow was observed when turbulent vortex occur in branch arm OY axis. Turbulent kinetic energy profile analysis on lines, parallel to OX, correspond to that of velocity while along distant from branch arm, k smoothly decreases but near the branch arm, and k appears the maximum values. This maximum corresponding to extremes values of velocity which occur in the beginning of branch arm OY axis.

The turbulent kinetic energy is represented at Figure 15. At the T-junction's neck, convective mechanism begin to play governing role, improving the flow separation from the wall. Outlet_1 is also significantly not symmetry. At the wall opposite the branch balance is quite to that of steady flow, area of flow attachment characterize with large scale perturbation of convective and turbulent diffusive component. In branch arm of OY axis, a very complicated structure turbulent kinetic energy balance was observed.

Furthermore, the fluid may also cause some effect on its turbulent kinetic energy due to the fluid velocity at the main arm OX axis is continuously undergoing changes in both magnitude and direction of the turbulent flow, As the Figure 15 represented below, the effect of turbulent kinetic energy is clearly happened at the beginning of branch arm OY axis where the velocity magnitude is maximum. The turbulent kinetic energy can be visualized of consisting irregular swirls of motion called eddies. Usually, turbulence consists of many different size of eddies define by turbulent spectrum. The Figure 16, 17 and 18 shows the difference visualization of turbulent kinetic energy.

The greater effect on turbulent kinetic energy is due to the greater continuous changes of fluid velocity. Figure 18 shows that a little effect on turbulence kinetic energy at that particular zone. Even the area mention has maximum velocity magnitude but due to little continuous changes of fluid velocity, the effect of turbulent kinetic energy is small.

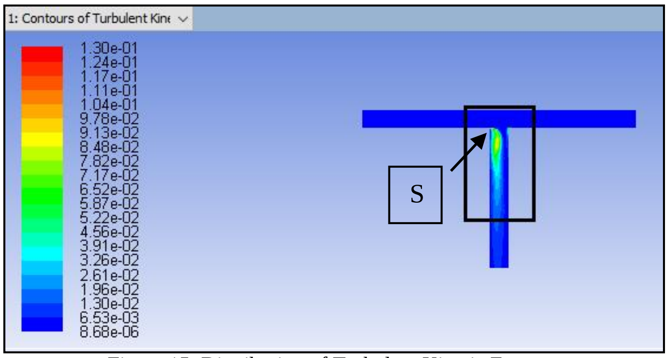


Figure 15: Distribution of Turbulent Kinetic Energy

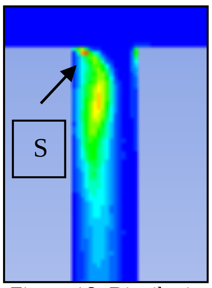


Figure 16: Distribution of Turbulent Kinetic Energy at High Velocity Flow

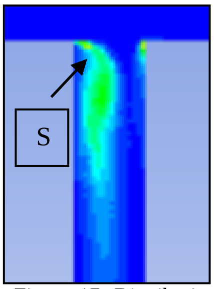


Figure 17: Distribution of Turbulent Kinetic Energy at Medium Velocity Flow

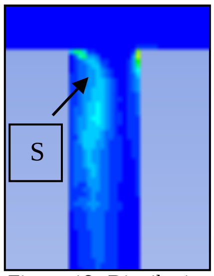


Figure 18: Distribution of Turbulent Kinetic Energy at Low Velocity

B. Effect of Fluid Structure Interaction (FSI) on the T-junction that causing to erosion

As we proved that effect of gas-liquid velocity, radial distribution of total pressure and turbulent kinetic energy are highly effected in the branch arm's neck OY axis where this location is called as constriction area. The visualization of contour gas-liquid velocity, radial distribution of total pressure and turbulent kinetic energy undergo changes at that particular area where the maximum gas-liquid velocity impact, high pressure of radial distribution and greater contour effect on turbulent kinetic energy.

Figure 19 illustrated the erosion rate at the early stage of pipeline T-junction. As the velocity of the fluid increases, the maximum erosion rate at the early branch arm also increases. The maximum erosion rate is relatively small due to the low velocity hit the internal wall which produced shear stress between the fluid and the pipe structure. This phenomenon called as liquid droplet erosion where formation of high pressure and stresses at that particular point produced liquid droplets contact with internal pipe wall. Generally, droplets erosion is a problem in a very high velocity even wet gas with an annular mist flow can cause droplets impact to happen.

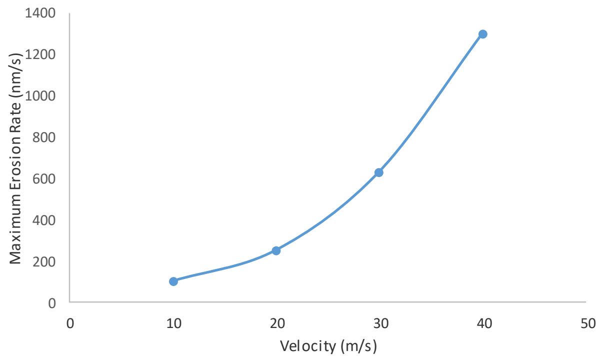


Figure 19: Maximum Erosion Rate in Velocity Varies

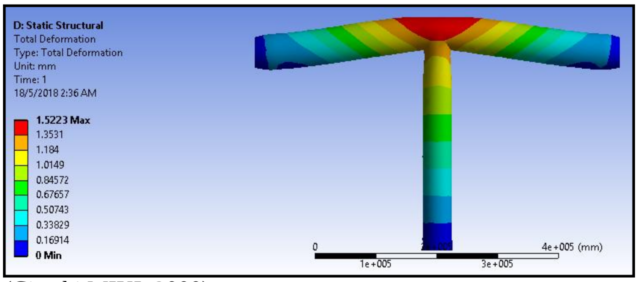
C. *Effect of pipe structure on Fluid Structure Interaction (FSI)*

In order to estimate the ultimate strength of structure steel pipeline T-junction, the structure of pipe bending either in closing bending or opening bending were studied. This is to ensure the optimization of pipe strain, pipe stress or pipe deformation of the structural steel T-junction which subjected to the pipe strength. Ultimate strength of steel pipe T-junction is studied with respect to initial crack to describe the allowable value pipe strength.

Due to the interaction of fluid domain of turbulent flow to the solid domain, it may also cause effects on pipe structure depending on their ability to withstand with high impact of the domain flow. To study the geometrical effect of pipe bending or deformation obtained on a pipe was analyzed by using ANSYS Static Structural Analysis. The deformation obtained on the pipe and the contour representing in Figure 20, 21 and 22. The deformation of pipe shows the closing mode bending, and end of the pipe was pull inward and downward.

Unloading the process of both internal pressure and applied load was introduced to conservatively ultimate strength. This is because the crack did not initiated even up to maximum displacement of capacity limit of loading but when the unloading process of both applied load and internal pressure was introduced, the crack initiated at the unloading process. The ovalization of bending and initiation of crack can be seen near central section of main branch arm OX axis.

Based on the above Figure 20, the deformation is more on the center of pipe T-junction. This is due to the maximum strain occurs at central section of structural steel. In this study, the ultimate strength of pipe deformed was defined as the ultimate state in accordance with crack initiation. The large axial compressive strains had been accumulated until when the crack is initiated. The deformation found to have similar trend as the investigated by



(Citoshi MIKI, 2000)

Figure 21: Deformation of Pipe at Low Pressure

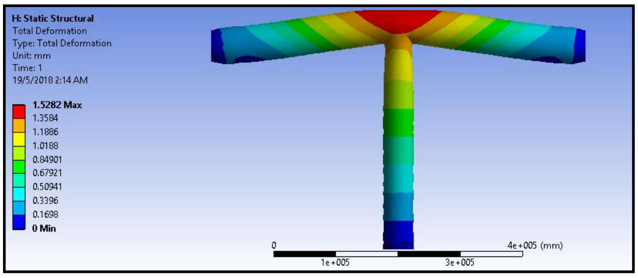


Figure 22: Deformation of Pipe at Medium Pressure

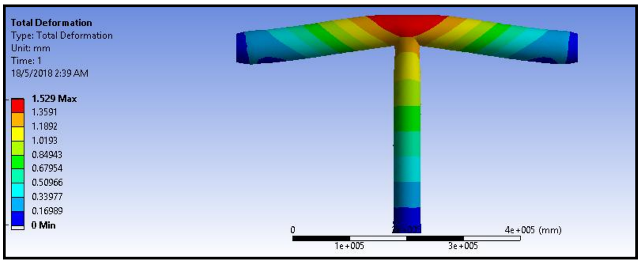


Figure 23: Deformation of Pipe at High Pressure

IV. CONCLUSION

Computational Fluid Dynamic (CFD) study has been conducted to investigate the phase separation and deformation of structural steel pipeline T-junction. Deformation of structural steel pipe and its effect on the fluid domain was also studied using Fluid Structure Interaction (FSI). A several conclusion were made based on the study by applying the boundary condition. By analyzing the multiphase flow at the T-junction, it can be seen that fluid flow undergo phase redistribution at the branch arm's neck of OY axis and the center of main arm OX axis where the location is opposite to the branch arm. This can be proved by considering the effect of fluid velocity, radial distribution of total pressure and effect of turbulent kinetic energy. From the contour of fluid velocity, fluid flow velocity undergo velocity disturbance at the center of main arms OX axis and beginning of branch arm OY axis. This area is called as constriction area. The total pressure radial distribution is highly effect on the constriction area which strongly prove that the phase redistribution occur at that area. Due to the fluid flow velocity effect, turbulent kinetic energy clearly happened. By considering the Fluid Structure Interaction that causing to erosion, based on analyzing the velocity impact, the most maximum erosion rate happened at the constriction area compared to other part of pipeline T-junction. Hence, the deformation of pipe is more at the center of pipeline T-junction due to maximum strain occur when introducing the load and internal pressure.

ACKNOWLEDGMENT

Firstly, I wish to thank Allah for giving me the opportunity to embark on my degree and for completing this long and challenging journey successfully. My gratitude and thanks go to my supervisor Assoc Prof Ir. Dr. Nadiahnor Bt. Md Yusop for the continuous support. Her guidance helped me all the time of research and writing this thesis. I could not imagined having a better advisor and mentor of my degree study. Second, I would like to express my sincere gratitude to my family especially my parent who always become the reason why I am here. I am nothing without their support and pray. Lastly, to my fellow friends, thank you for the mini simulation discussion and opinion for completing this challenging journey.

NOMENCLATURE

FSI		Fluid Structure Interaction
CFD		Computational Fluid Dynamic
BPG		Best Practice Guideline
ρ	[kg/m^3]	Density fluid
U	[m/s]	Velocity vector of the fluid
t	[s]	Time
α_g	[-]	Volume fraction of gas
α_L	[-]	Volume fraction of liquid
g	[N]	Gravitational force
U_g	[m/s]	Superficial gas velocity
U_L	[m/s]	Superficial liquid velocity
U_t	[m/s]	Total superficial velocity
n	[-]	Number of phase present in the fluid
u_i	[m/s]	Average phase velocity
x_i	[-]	Flow quality
m_i	[kg s^{-1} m^{-2}]	Mass flux of ith phase
V_w	[m/s]	Velocity of water
V_a	[m/s]	Velocity of air

REFERENCES

[1] Athulya, A. S., & Miji Cherian, R. (2016). CFD Modelling of Multiphase Flow through T Junction. *Procedia Technology*, *24*(Icetest 2015), 325–331. https://doi.org/10.1016/j.protcy.2016.05.043

[2] Chen, J., Wang, Y., Li, X., He, R., Han, S., & Chen, Y. (2015). Erosion prediction of liquid-particle two-phase fl ow in pipeline elbows via CFD – DEM coupling method, *275*, 182–187. https://doi.org/10.1016/j.powtec.2014.12.057

[3] Comsol.com. (2017). *What Is Fluid-Structure Interaction?*. [online] Available at: https://www.comsol.com/multiphysics/fluid-structure-interaction.

[4] Granger, R. (2012). *Fluid Mechanics*. Dover Publications.

[5] Liu, S., Sang, S., Ma, J., Wang, X., Wang, W., & Du, Y. (2017). Journal of Petroleum Science and Engineering, *148*(September 2016), 21–31. https://doi.org/10.1016/j.petrol.2016.10.006

[6] Rzehak, R., & Kriebitzsch, S. (2015). Multiphase CFD-simulation of bubbly pipe flow: A code comparison. *International Journal of Multiphase Flow*, *68*, 135–152. https://doi.org/10.1016/j.ijmultiphaseflow.2014.09.005

[7] Siva, V., & Sankar, P. V. (2015). Fluid Structure Interaction Based Investigation of Convergent-Divergent Nozzle, *5*(5), 141–143

[8] Thermopedia.com. (2017). *Bubble Flow*. [online] Available at: http://www.thermopedia.com/content/8/.

[9] Thermopedia.com. (2017). *Multiphase Flow*. [online] Available at: http://www.thermopedia.com/content/4/.

[10] Thermopedia.com. (2017). *T-junctions*. [online] Available at: http://www.thermopedia.com/content/268/.

[11] Vasava, P. R. (2007). Fluid flow in T-junction of pipes, 75. Retrieved from http://www.uvm.edu/~pdodds/teaching/courses/2009-08UVM-300/docs/others/2007/vasava2007a.pdf