IOP Conference Series: Materials Science and Engineering

PAPER • OPEN ACCESS

The Effect of Flow Variations to Vibration Tendency in A Hydraulic Manifold

To cite this article: Norazhar Ali *et al* 2020 *IOP Conf. Ser.: Mater. Sci. Eng.* **884** 012113

View the article online for updates and enhancements.

IOP Conf. Series: Materials Science and Engineering **884** (2020) 012113 doi:10.1088/1757-899X/884/1/012113

The Effect of Flow Variations to Vibration Tendency in A Hydraulic Manifold

Norazhar Ali[1], Kahar Osman, Fazila Mohd Zawawi and Muhammad Noor Afiq Witri Muhammad Yazid
[1]School of Mechanical Engineering, Universiti Teknologi Malaysia, Johor, Malaysia.

E-mail: norazharali@gmail.com, kaharosman@utm.my,fazila@mail.fkm.utm.my, mnafiqwitri@utm.my

Abstract.A hydraulic manifold is an important component in hydraulic machinery to convey hydraulic oil under high pressure into hydraulic tube and hose for cleaning purpose. Due to high pressure during operation, the process induces vibration and leakage can occur at the exit ports of hydraulic manifold. The aim of this study is to determine the effects of pressure and velocity variations in hydraulic manifold with respect to vibration tendency. In this study, computational fluid dynamics (CFD) is used to simulate the hydraulic manifold fluid behaviours, under various operating conditions. The result shows that fluctuations in pressure and velocity occurs at each branch in a mainstream due to changes of the area and geometrical shape. We also observed surge of pressure but reduction in velocity at each branch. Overall, the results show the most affected areas are those near the inlet. Areas further downstream are not significantly changed by the increments.

1. Introduction

A hydraulic manifold is a device to distribute the fluid flow to channels with certain pressure and mass flow rate. it is widely used in the mechanical. Chemical, biomedical, civel and environmental engineering. Common type of hydraulic manifold are divided into 4 types which dividing manifold, combining manifold, paraller (Z-type) and reverse (U-type)[1].

In this research, dividing flow is the type of hydraulic manifold. The purpose of hydraulic manifold is used for cleaning or flushing the debris inside the tubes and hoses under certain pressure that create high velocity. Debris trap inside the tube and hose able to clogged the system and affect the hydraulic components which reduce the response and life expectancy.

In this research the physical of hydraulic manifold is identical to the actual used in industry. The hydraulic manifold made by carbon steel grade S275JR. The cavity machined by conventional drilling machine with a constant diameter. At the end of each port, NPT threaded is machined in order to assembly the hydraulic fitting.

The hydraulic power unit is used to generated pressure in the process of flushing. The pressure regulate by regulator in order to control the vibration of the hydraulic manifold and system. The process of flushing is closed loop. The performance of flushing under pressure shall be vibration free in a hydraulic manifold which affect quite process, minimum pressure drop, flushing efficiency and leaks free.

A number of investigation of flow distribution and behaviour in a hydraulic manifold have been done such as theoretical analysis, experimental and simulation. CFD simulation method is to predict the performance and efficiency in a hydraulic manifold. In this study, flow distribution and behavior, pressure and velocity are the critical subjects to simulate in a hydraulic manifold. Flow distribution and pressure drop effects by the geometrical shape and variations of pressure and velocity caused vibration and noise [1][3]. The branch effects unpleasant flow in cavity and lead to mechanical damage.

The fluid observed leakage at the connection between the hydraulic manifold and fitting. This event occurs over period of time. Probability fluid induced vibration happen at the outlet channel caused the fitting loosen or damage the ferrule or seal. Moreover the hydraulic manifold vibrated and noise due to lack of support. Therefore, the flow distribution problem exist in a hydraulic manifold probably by number of outlets, geometrical size and shape, flow area ratios and channel length.

The objective of this study is to determine the effects of pressure and velocity variation in hydraulic manifold with respect to vibration tendency. The changes of pressure and velocity lead the fluid induced vibration flow in hydraulic manifold. The study covered pressure and velocity variation in cavity particularly in mainstream, branches and outlets. Therefore, commercial software ANSYS Fluent and geometry of cavity sketch by Solidwork to simulate the flow investigation. The industrial parameters such as pressure and velocity will be used in this simulation. The vibration tendency will be locate and identify the causes based on pressure and velocity in a cavity.

2. Methodology

2.1. Physical Model

The hydraulic manifold studied here was a flow distribution channel used hydraulic fluid in flushing system. The system consists of dividing manifold which one inlet and 6 outlets with constant cross-section as shown in Figure 1. The model sketch by Solidwork and convert to .step file for ANSYS to read and process it.

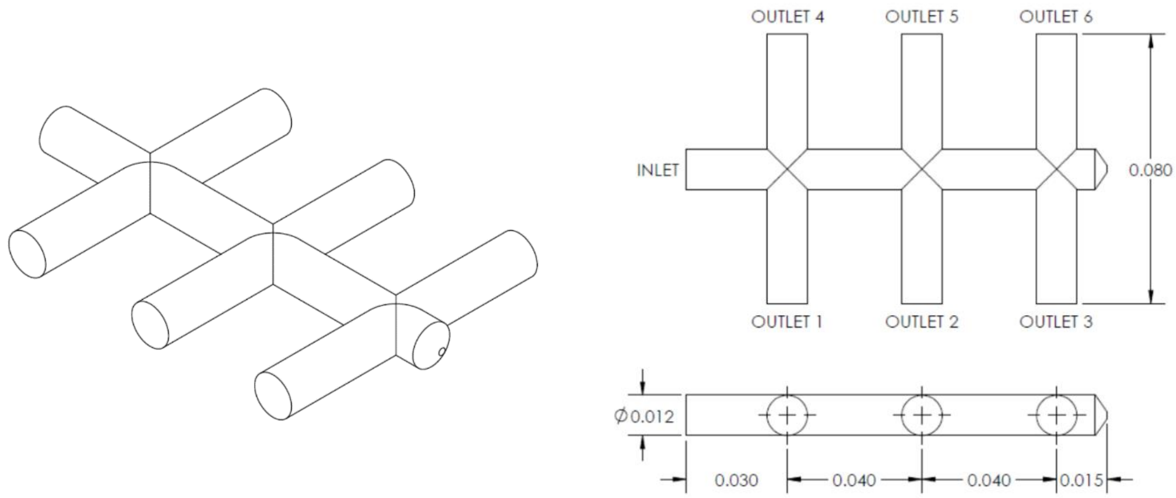

Figure 1: Geometry of hydraulic manifold

2.2. Numerical Method
The numerical simulation is based on the actual operation of flushing process in industrial as following below:

1. Hydraulic fluid Shell Tellus S2 M32 as working fluid and fluid flow is incompressible and single-phase turbulent flow.
2. Inlet and outlet have the same sizes of diameter.
3. Fluid properties kept constant (Temperature 40^0C, density 0.875kg/l and viscosity 32 cSt).

ANSYS Fluent was employed to solve the simulation in a finite volume method and the second-order upwind scheme was adopted for equation discretization. The realizable of k-emodel is chosen as the turbulence model. The solution method of pressure velocity-coupling schecme is SIMPLE with second order spatial discretization with high order term relaxation. The residual monitoring of continuity specified to 10^{-4} for better convergence for all variables.

2.3. Boundary Conditions
The inlet was defined as the pressure inlet. While the outlets set as pressure outlets and the gauge pressure was set to 0 gauge (Pa) which equal to atmospheric pressure. The pressure inlet set at $120x10^5$ Pa.

3. Verification and Validation

3.1. Grid Independence Study
The grid independence study was performed with 4 grid system. Refer to Table 2 for the detailed of grid arrangement. The refinement was conducted in the near wall body. The grid evaluation was based on the point in the axial centreline (line 1) of the hydraulic manifold mainstream as shown in Figure 2. The evaluation point taken at 0.031m at the middle of branch 1. Grid generation and refinement of hydraulic manifold at 0.5mm grid size.

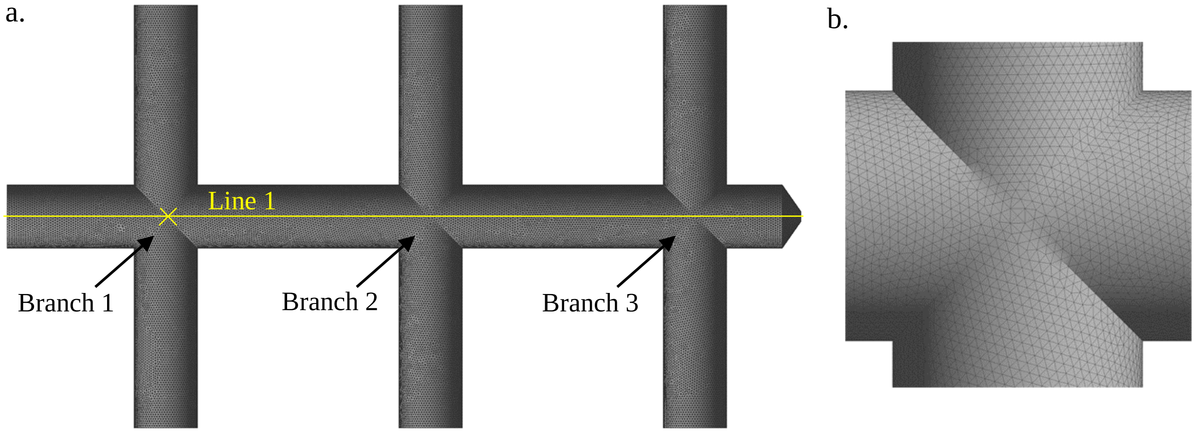

Figure 2: a. Point at mainstream line in middle of hydraulic manifold, b. Mesh structure

Table 1: Grid independence study result

No	Grid size, mm	Mesh size	Pressure, Pa	Velocity, m/s
1	0.7	1,137,927	$11.71x10^5$	200.05
2	0.6	1,790,586	$11.55x10^5$	200.18
3	0.5	3,029,739	$11.24x10^5$	200.35
4	0.4	5,764,900	$11.09x10^5$	200.45

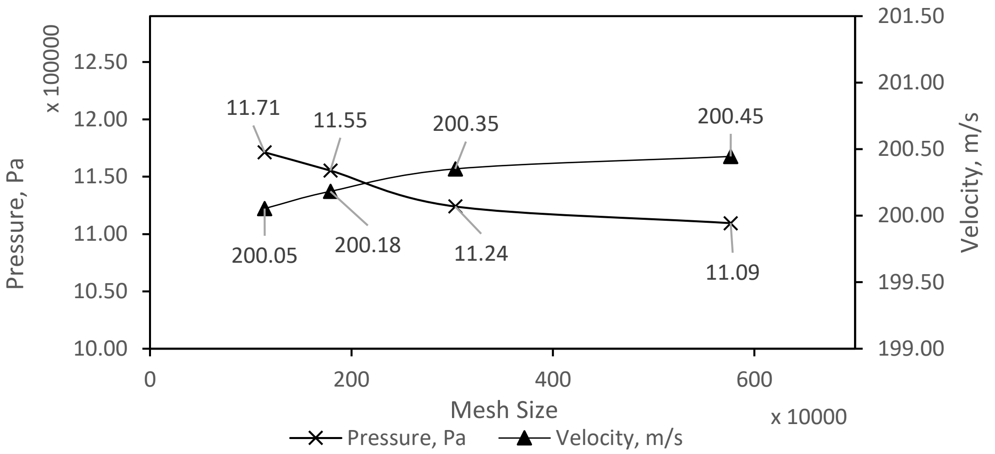

Figure 3: Graph of grid independence result

A slight variation was observed between grid 0.5 and 0.4. refer to the graph plotted in Figure 3, pressure is dropped after reduction of grid size while velocity is increased. The difference between grid 0.5 and 0.4 is only 1.29% in pressure and 0.05% in velocity. Therefore, the grid size 0.5 was chosen as the indepence grid size.

The verification of the result compared to numerical prediction in 90^0bend system in a circular cross-section [5]. The hydraulic manifold is characterized by a velocity pattern in cross-section center line as shown in Figure 4(a). The velocity map and profile is similar in flow pattern. Even though, the geometry of the channel is different but it is owing to the variation in the relative magnitude of frictional losses near to wall body and fluid flow in the center.

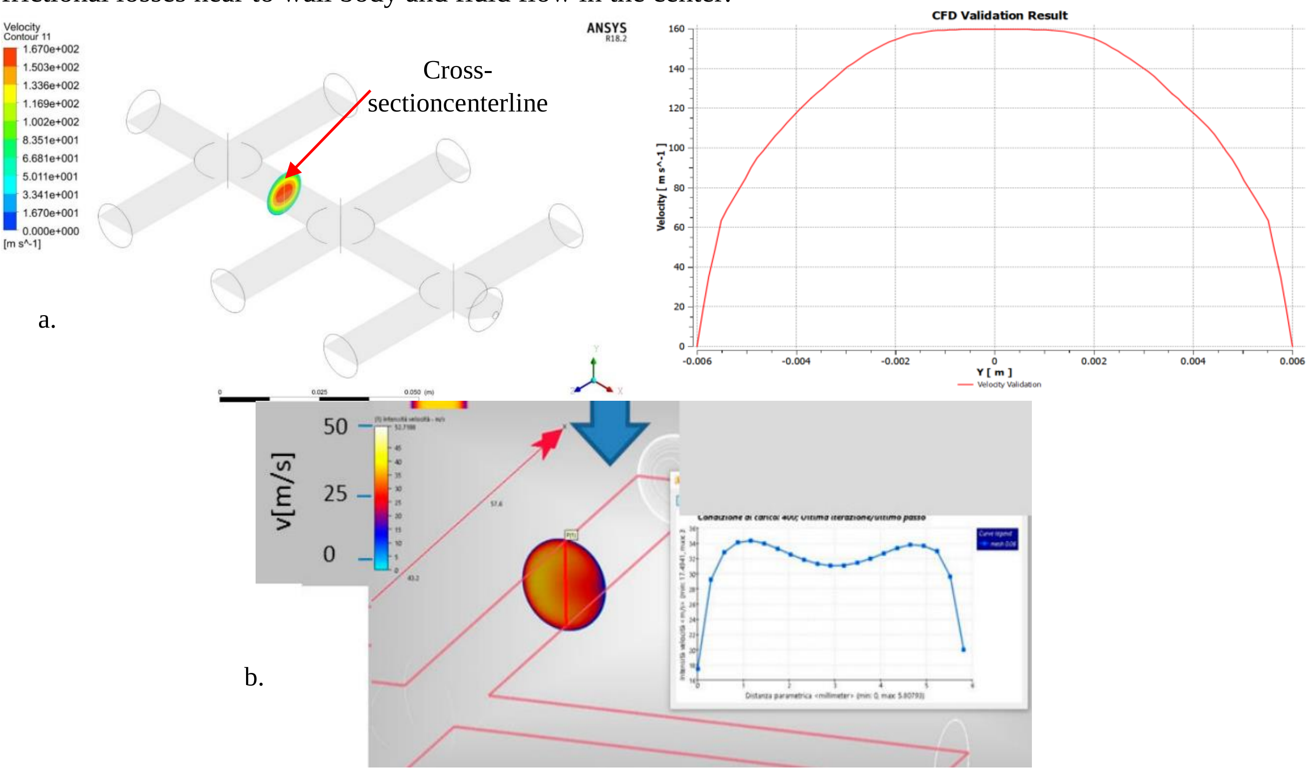

Figure 4: Computational fluid dynamic verification; a. Hydraulic manifold, b. Past literature: 90^0 elbow numerical study

Pattern comparison between research and literature review [8] on the pressure fluctuated at branches as shown in Figure 5. The authors state that the pressure shows a sharp decline in the junction center, then the pressure increased at the entrance section of the branch tube. Then the pressure decreased along the direction of fluid flow. This is a similar pressure fluctuation behaviour for the research.

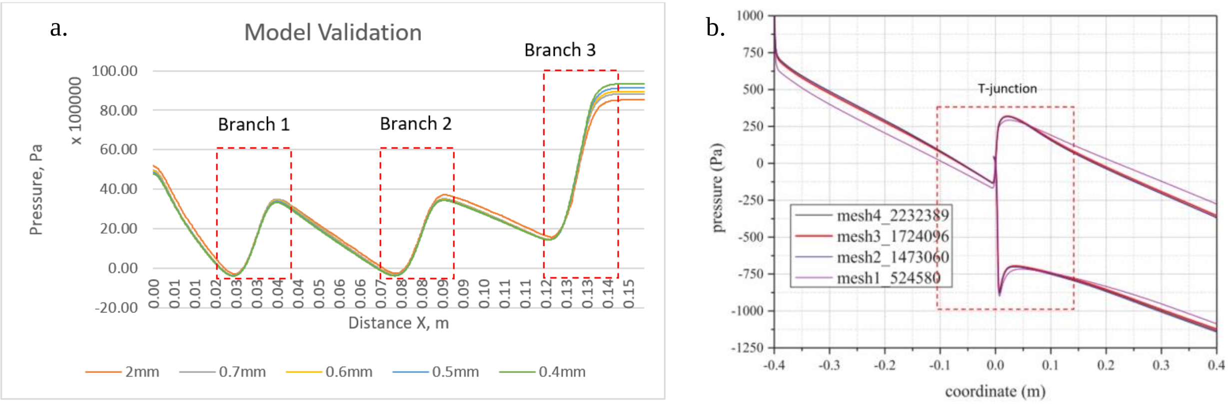

Figure 5: Pressure drop patterns in a. Branches, b. T-junction

4. Results and Discussion

4.1. Streamline

Flow distribution in streamline 3D as shown in Figure 6(a), high velocity on mainstream that affect the flow at outlet 1 and 2 with low velocity and disorganizeflow compare to outlet 3. The velocity decreases after the branch 1 and the velocity regain when approaching branch 2. Then the velocity decompasses at branch 3 and flowing out through the outlet 3. The fluid flow dissipation and develop chaotic of flow due to branch geometry at outlet 1 and 2. Moreover the distance between the branches gave an effect to this matter.

Figure 6 (b) shows the streamline flow pattern shown in the plane X-Z which is a cross section of the hydraulic manifold cavity in top view. Refer to the figure at branch 1 and 2, the fluids flowing through it seem disorganized due to high velocity in a mainstream and create swirl flow and back stream. Outlet 1 shows the largest swirl flow compared to outlet 2 and outlet 3 shows a very small swirl flow and back stream.

Figure 6 (c) shows the streamline flow patterns for outlet 1, 2 and 3 based on plane Y-Z. From this figure, the 4 sets of swirl flow happened at each outlet 1 and 2 which is near to inlet. However, only low velocity measuredat the outlet 3.

IOP Conf. Series: Materials Science and Engineering **884** (2020) 012113 doi:10.1088/1757-899X/884/1/012113

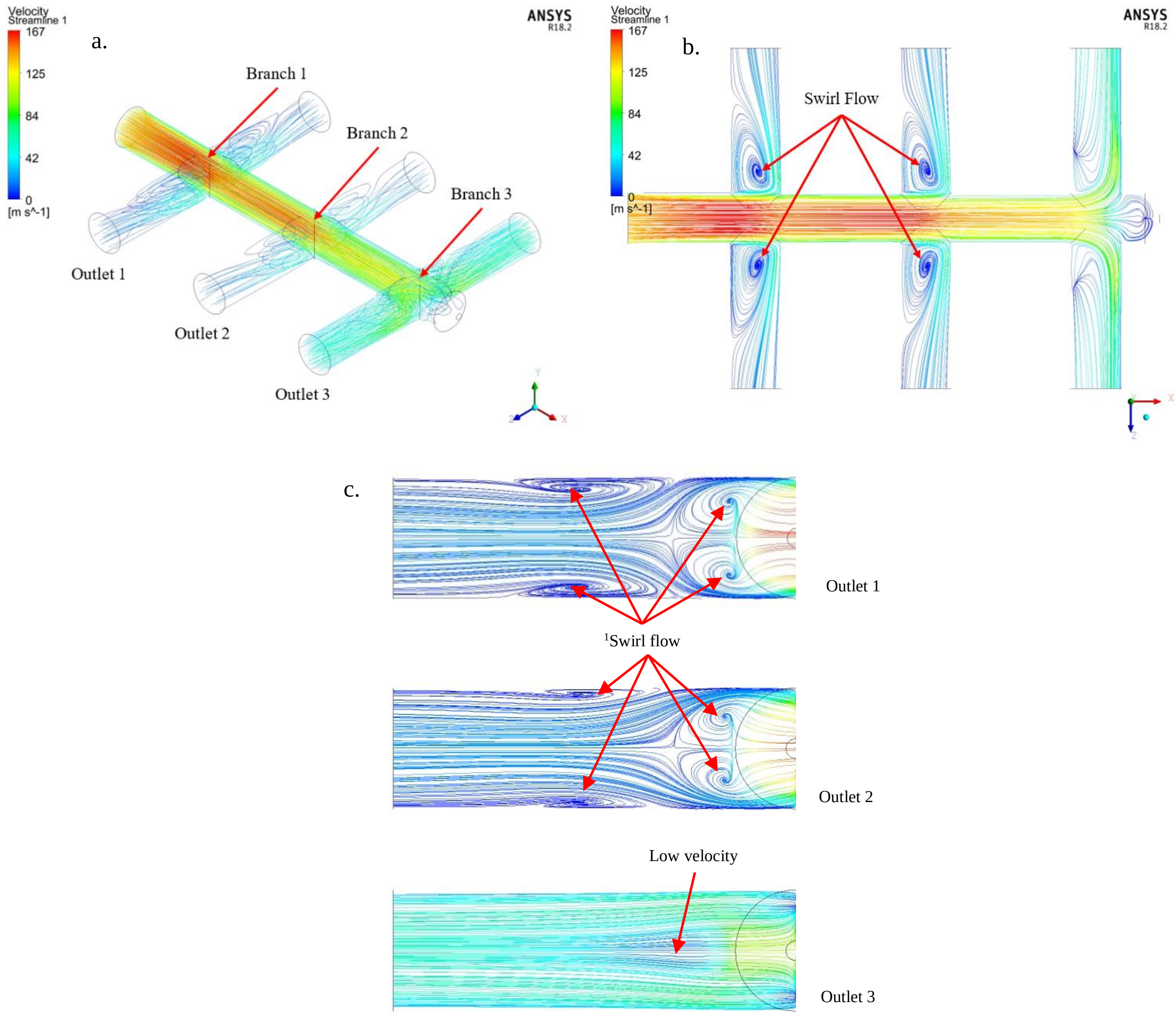

Figure 6: Streamline; a. 3D view, b. Plane X-Z and c. Plane Y-Z for outlet 1, 2 and 3

4.2. Mainstream: Pressure-velocity

Velocity measured at inlet is 140ms^{-1} based on pressure inlet as initial boundry condition which is 120x10^{5} Pa. Outlet 1 to 3 set as pressure outlet at 0 gauge (Pa). Refer to Figure 7, the fluctuation of pressure and velocity happened after the branch, particularly branch 1 and 2. This is due to changes of geometry and cross-sectional area.

Due to high velocities at mainstream, swirl flow and back stream occur at the entrance of outlet 1 and 2 that affect the pressure drop to negative and low velocity. However, at outlet 3, dissipation of fluid flow at the end of branch cause the flow distribution better with positive pressure and higher velocity. The swirl flow and back stream confirm the statement of vibration tendency at outlet 1 and 2.

Negative pressure measured at the branch 1 and 2 entrances and along the outlet 1 and 2 channel. The negative pressure develops a cavitation that lead to vaporize and bubbles of hydraulic fluid. Therefore the vibration tendency to happen is in that location. Moreover the sudden drop of pressure or can measured in pressure differential also the cause of vibration according to the past literature review.

Pressure and velocity changes are similar trend and pattern which pressure drop and velocity increase at branches. The change of the area in cross-section is affected the pressure-velocity variation. Increasing pressure to maximum at the stagnation point which end of the hydraulic manifold cavity with no flow.

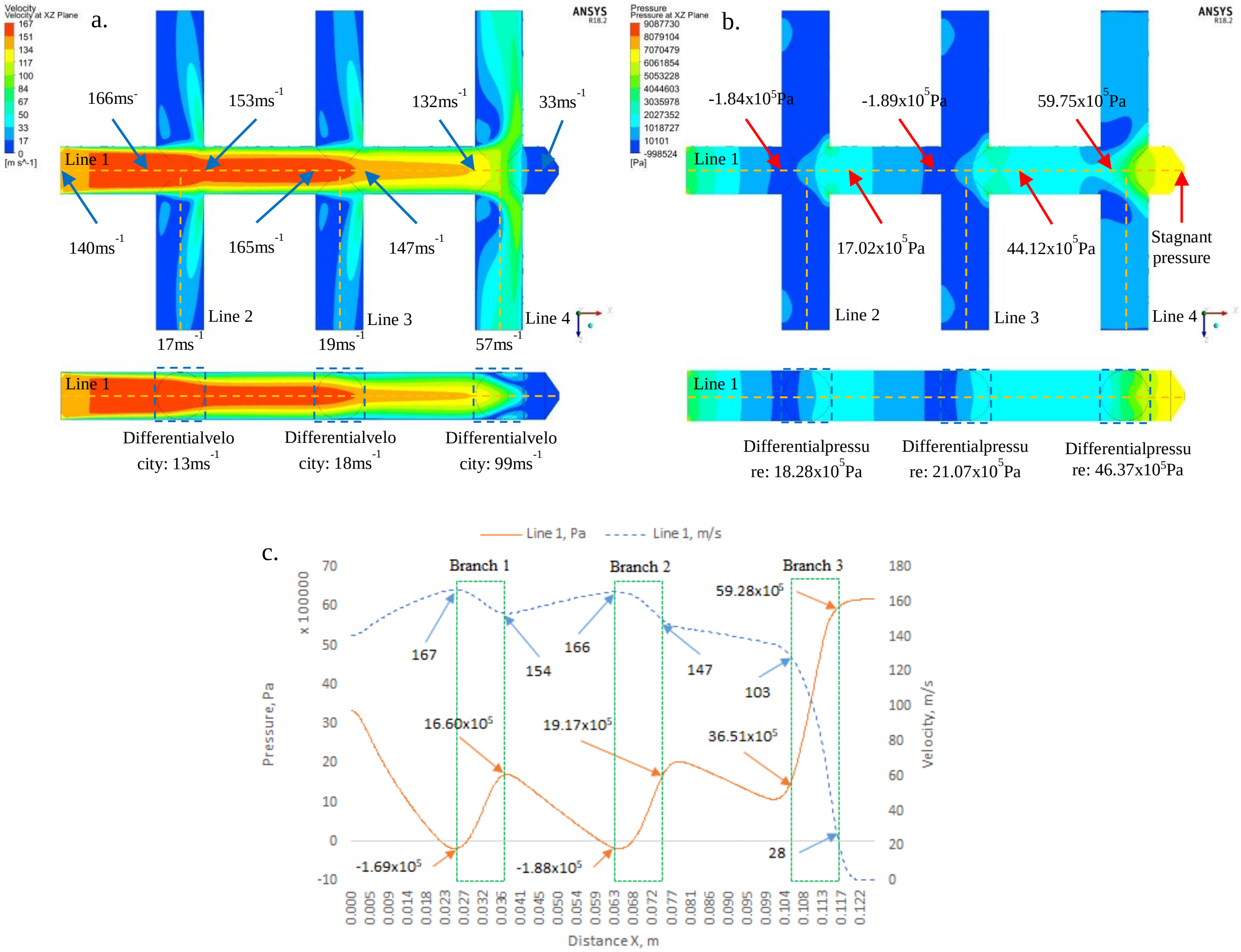

Figure 7: Mainstream; a. Velocity, b. Pressure and c. Graph of pressure-velocity

4.3. Outlets: Pressure-velocity

Refer to Figure 8 (a) and (b), Pressure-velocity variation at outlet 1 and 2 shows the pressure and velocity drop rapidly and develop a high differential of pressure and velocity. Plus, the behavior of the pressure and velocity is similar. Outlet 3 shows higher in pressure and velocity which lead to better flow distribution. According to past literature review, the maximum mass flow rate achieved at the end of the outlet [1, 2]. There is more fluid flowing through the last outlet with higher velocity.

Developing of negative pressure and low velocity the outlet 1 and 2 channel tendency of vibration highly occur at this position. Moreover the distance gap of negative pressure outlet 1, 2 longer than outlet 3. However outlet 3 show positive pressure and higher velocity that lead to better fluid flowing through the outlet 3.

IOP Conf. Series: Materials Science and Engineering **884** (2020) 012113 doi:10.1088/1757-899X/884/1/012113

From graph plotted shown in Figure 8 (c, d and e), the red box highlighted shows the negative pressure region with lower velocity particularly critical for outlet 1 and 2. The negative pressure measured in cavity indicate the cavitation effects on that region. The cavitation is lead to the vibration tendency and possibility of a bubble and vaporized of fluid.

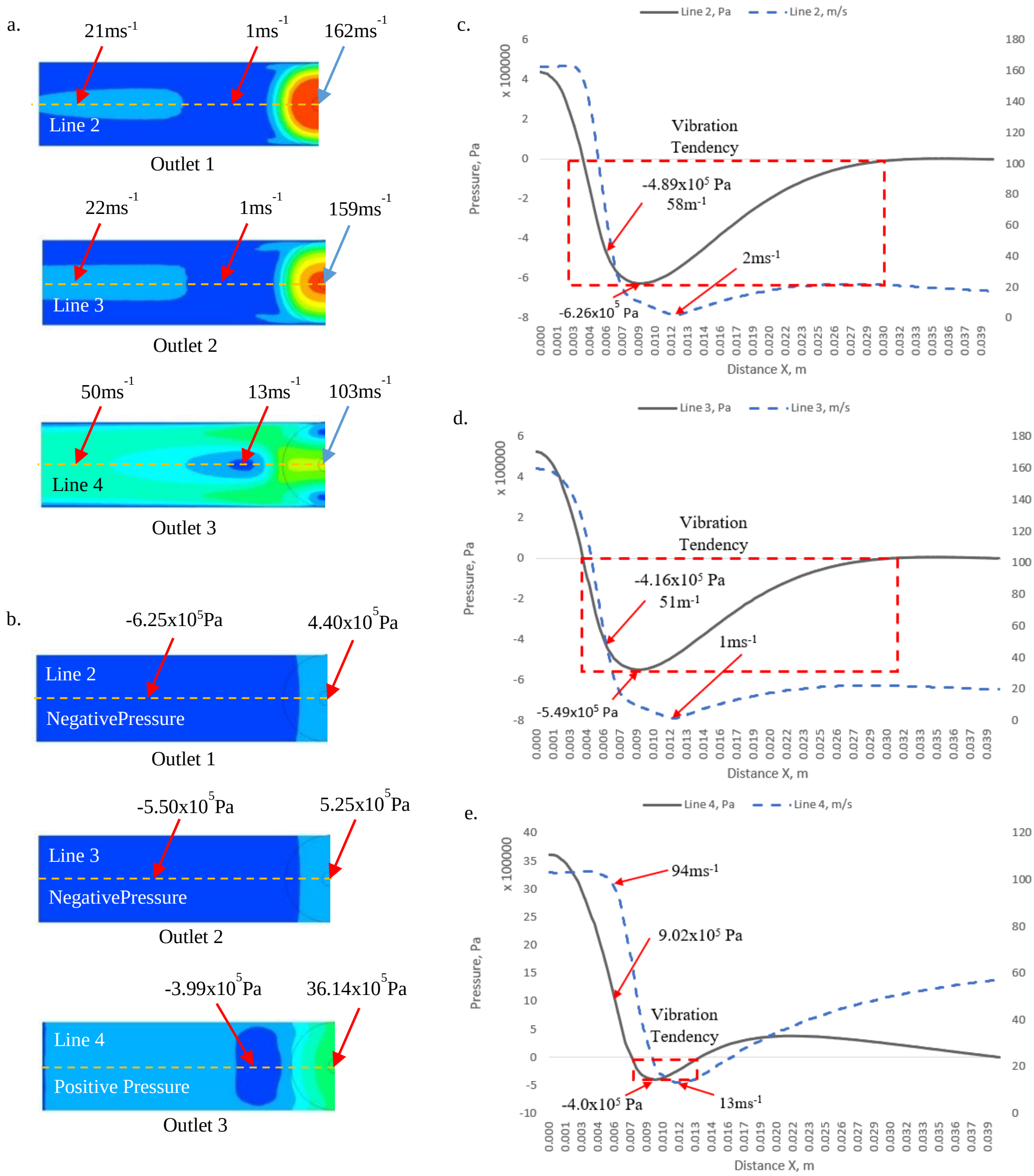

Figure 8: Outlets; a. Velocity, b. Pressure, c. Graph pressure-velocity outlet 1, d. Graph pressure-velocity outlet 2 and e. Graph pressure-velocity outlet3

5. Conclusion
As conclusion, the variations of pressure and velocity affects the vibration in a hydraulic manifold cavity. Based on our computed results, the vibration tendency occurs due to differential pressure and velocity, negative pressure, low velocity, swirl flow and back stream in the hydraulic manifold cavity.Pressure is increasedwhile velocity is decreased at each of branch in a mainstream. This is due to geometrical changes in hydraulic manifold cavity. Branch 1 and 2 shows the same behavior except branch 3. The negative pressure found before the entrance of branch 1 and 2 with high velocity. The pressure increased in the middle of branch 1 until branch 2 entrance. However, positive pressure measure at branch 3 with better flow distribution.Due to the phenomenon described above, there is tendency of vibration to occurs at branch 1 and 2 and outlet 1 and 2. Refer to Figure 9 shows the location of vibration tendency inside the cavity.

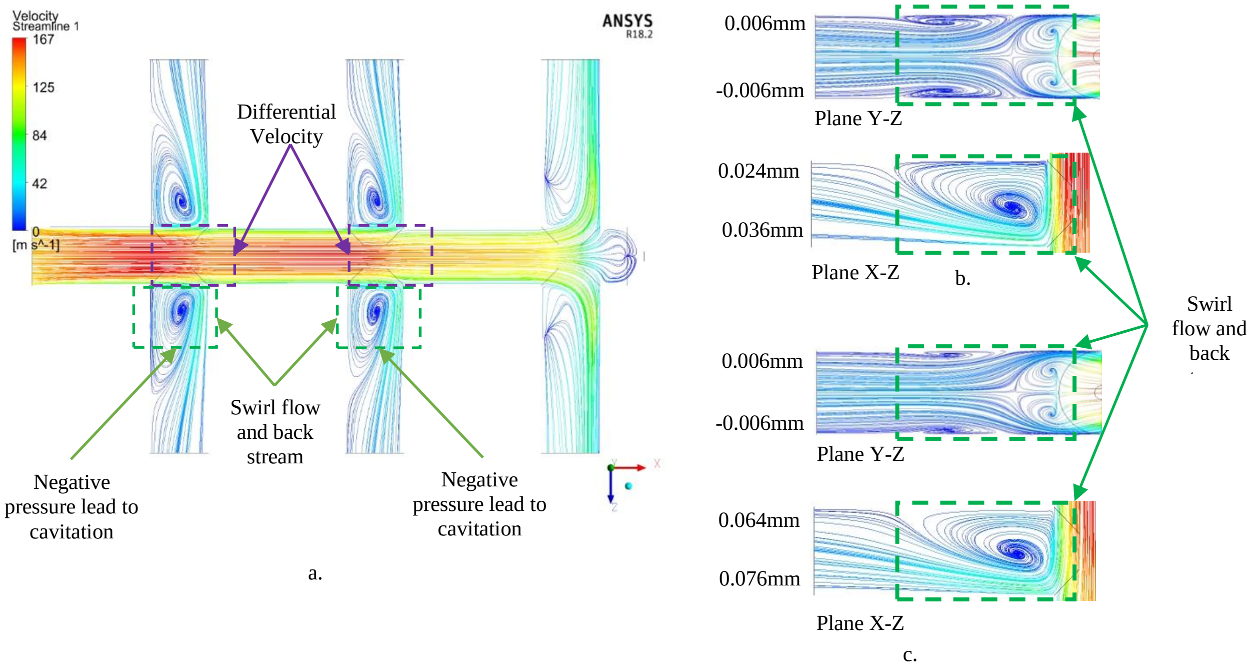

Figure 9: Vibration tendency; a. Plane X-Z, b. Outlet 1 and c. Outlet 2

References
[1] Wang, J.(2011). Theory of flow distribution in manifolds. Chemical Engineering Journal, 168(3), 1331–1345.
[2] Zhang, W., Li, A., Gao, R., & Li, C. (2018). Effects of geometric structures on flow uniformity and pressure drop in dividing manifold systems with parallel pipe arrays. International Journal of Heat and Mass Transfer, 127, 870–881.
[3] Lisowski, E., & Raida, J. (2013). CFD analysis of pressure loss during flow by hydraulic directional control valve constructed from logic valves. Energy Conversion and Management, 65, 285–291.
[4] Kushwaha, T. N. (2016). CW pump fluid induced vibration troubleshooting methodology. Procedia Engineering, 144, 274–282.
[5] Zardin, B., Cillo, G., Rinaldini, C. A., Matarelli, E., & Borghi, M. (2017). Pressure losses in hydraulic manifolds. Energies 10, 310.
[6] Siba, M., Wanmahmood, W., Zaki Nuawi, M., Rasani, R., & Nassir, M. (2016). Flow-induced vibration in pipes: Challengess and solutions - A review. Journal of Engineering Science and Technology, 11(3), 362–382.

[7] Moreno, C. J. G. (n.d.). Characterizing the bistable flow, of BWR, as a bifurcation (pitchfork type) in the Navier - Stokes' equation solution., (86), 1–12.
[8] Dinardo, G., Fabbiano, L., Vacca, G., & Lay-Ekuakille, A. (2018). Vibrational signal processing for characterization of fluid flows in pipes. Measurement: Journal of the International Measurement Confederation, 113, 196–204.
[9] Simão, M., Mora, J., & Ramos, H. M. (2014). Dynamic behaviour of a pipe system under unsteady flow and structure vibration, V Conferência Nacional de Mecânica dos Fluidos, Termodinâmica e Energia MEFTE 2014, Porto, Portugal, 11–12.
[10] Chenxiao, N. (2012). Study on Vibration and Noise for the Hydraulic System of Hydraulic hoist, Proceedings of the 1st International Conference on Mechanical Engineering and Material Science, 126–128.
[11] Martins, N. M. C., Soares, A. K., Ramos, H. M., & Covas, D. I. C. (2015). Wall shear stress in unsteady flow analysis using CFD. 12th International Conference Pressure Surges 2015 - Pressure Surges, Fluid Transients & Water Hammer, (November), 83–97.