

Numerical analysis and CFD simulation of fluid flow in T- Junction pipe By using ANSYS CFX

¹Netsanet Anteneh Ferede

¹Faculty of Engineering , Department of Mechanical Engineering,

¹University of Debrecen, H-4028 Otemeto ut 2-4, Hungary

¹netsanetanteneh79@gmail.com , ¹netsanet@mailbox.unideb.hu

Abstract

This Project deals with the fluid flow in the T-junction with the 90° angle of the adjacent branch and horizontal pipe having 100 mm diameter and the vertical branch 10 mm diameter. The flow is organized as a combining flow. It means that there are two inlet branches and only one outlet branch. There are two possible kinds of the flow combinations for this T-junction. The numerical Analysis and the CFD simulation in ANSYS were done for both flow combinations. Meshing has been made through the default mesh and inflation to each different diametrical pipe. Pressure loss, head loss, drag coefficient and the deviation from the numerical analysis (Error). Residual and its comparison between the numerical analysis for temperature pressure and velocity is presented in this project. Velocity distribution and average velocity profile is presented details of CFD simulation calculations and numerical analysis is done.

Key words: T junction, pressure loss, drag coefficient, Head loss, Bernoulli's equation

1.Introduction

The T-junction is used for the division or the combination of a fluid flow. The case of the flow combination is often used for mixing of two different phases. It is necessary to define a mathematical model of the T-junction and its coefficients in case of the T junction handling. The simplest T-junction mathematical model assumption that the pressures at the ends of pipes, forming the T-junction, are the same. There are some losses in the T-junction. Therefore, the more sophisticated description of the T-junction had to be derived [1]. It means they cannot be classified as loss coefficients. Another improved description of T junction was presented in the paper presented by Oka [2]. The total coefficients can be evaluated by the experiment or by the CFD computations. The goal of this project is to introduce the comparison between measurements and CFD computations of fluid flow in the T-junction with 90° angle of adjacent branch for case of combining flow. The combining flow in the T-junction means that there are two inlet branches and only one outlet branch.

2. Problem specification

In Figure1 it shows a schematic representation of the flow distribution through pipe and a general physical setup. Mercury enters the pipe at inlet 1 and inlet 2

and exit from outlet at 3. To analyze the fundamental system properties and flow patterns, a simplified flow model was employed. In this project to represent the pipe flow situation as the flow in a straight pipe i.e. main pipe with one vertical pipe i.e. branch pipe which is connected with the main pipe by an angle of 90°. The modeled flow distribution system considered for this purpose is of diameter D1 as 100mm for horizontal pipe or the main pipe and the vertical pipe considered id D2 as 10mm i.e. branch pipe. Mercury enters at inlet with T1=90°C and T2=30°C. The given parameters are listed in the next Table1.

The distribution pipe is modeled as an ideal 3D pipe with two inlets and one exit. The theoretical relationship for flow at pipe junctions has been developed by combining the conservation equation of mass, conservation equation of momentum and Bernoulli's principle to a control volume of the flow across a pipe junction [3]. Applying the Bernoulli's equation between upstream inlet and downstream outlet of a control volume, the diameter of the main pipe remains the same across the junction yields. Delta y, velocity distribution and average velocity, Drag coefficient [4], pressure drop and head loss is calculated both in numerical analysis and CFD simulation in ANSYS.

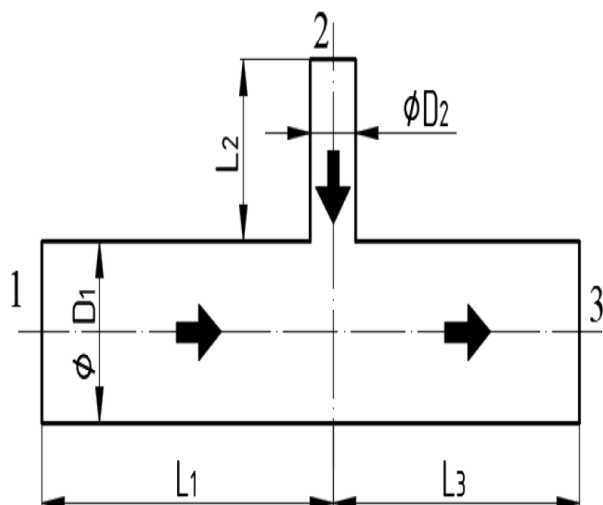


Figure 1; Schematic representation of the T-Junction

3. Boundary conditions for numerical analysis (Flow parameters)

3.1 Geometry

The geometry of fluid domain of T junction is created on ANSYS 19.2 CFX to study the Combination fluid flow phenomenon occurring on fluid when fluid flow enters a T junction from separate inlet.[5] The cross-sectional area of inlet run arm was named as inlet1 and the cross-sectional areas of branch arm is named as inlet 2 and the out let part of run arm was named as outlet and geometry of pipe is shown in Figure 2. The structure schematic drawing of T-junction is shown as Fig 2.

The inner diameters D1 of the main inlet pipe and outflow pipe are the same as 100 mm, and the inner diameter D2 of branch pipe is 10 mm. the length of each straight, $L1=L2=L3= 1000\text{mm}$.The T-junction is drawn with the Cartesian coordinate system. The fluid used in the simulations is mercury with density of 13546 kg/m^3 and dynamic viscosity of 0.001522kg/m s . The two inlets depend on the value of velocities and Temperature. The inlet boundary conditions are normal to surface area of inlet 1 and inlet 2. The velocity at the inlet pipe (upstream) is fully developed. It is assumed that no-slip boundary conditions at all the walls. As the flow is asymmetric the complete geometry is taken into consideration.[6]

Table 1 ;Given parameters

Material	Hg		
T1[°C]	90	L1[mm]	1000
T2[°C]	30	L2[mm]	1000
P1[Pa]	?	T3[°C]	?
P2[Pa]	?	P3	0
V1[m s ⁻¹]	2	V3[m s ⁻¹]	?
V2[m s ⁻¹]	25	D3[mm]	100
D1[mm]	100	L3[mm]	1000
D2[mm]	10		

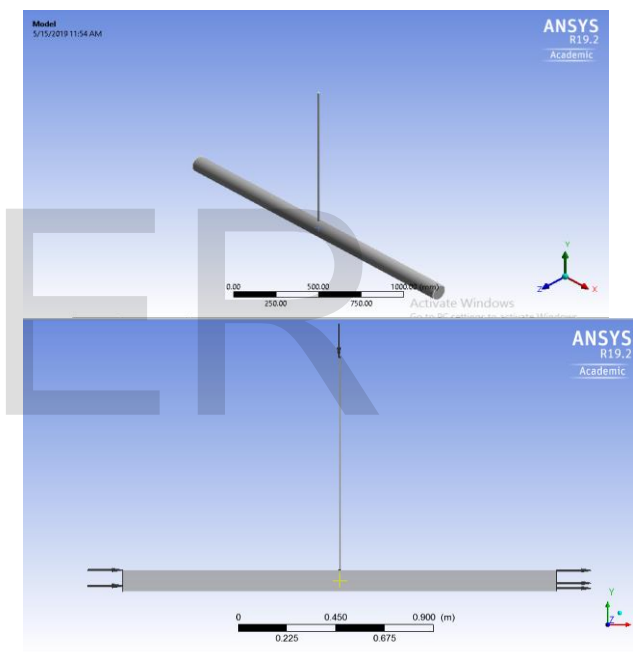


Figure 2 : 3D Geometry of T- junction pipe

3.2 CFD Mesh Model

To establish the CFD mesh model, the flow zone is divided into several subdomain for meshing. The region around tee junctions is meshed with tetrahedrons element for its good adaptability to the unstructured model, and hexahedron elements are generated in other zones to reduce the number of elements. Since there is two different diametrical pipe two inflation for meshing is used. the tetrahedrons element is much finer and the number of

elements is determined with checking computations.[7] The mesh model shown in Fig 3 includes 472324 nodes and 218870 elements. Details of the mesh information is located at the annexes.

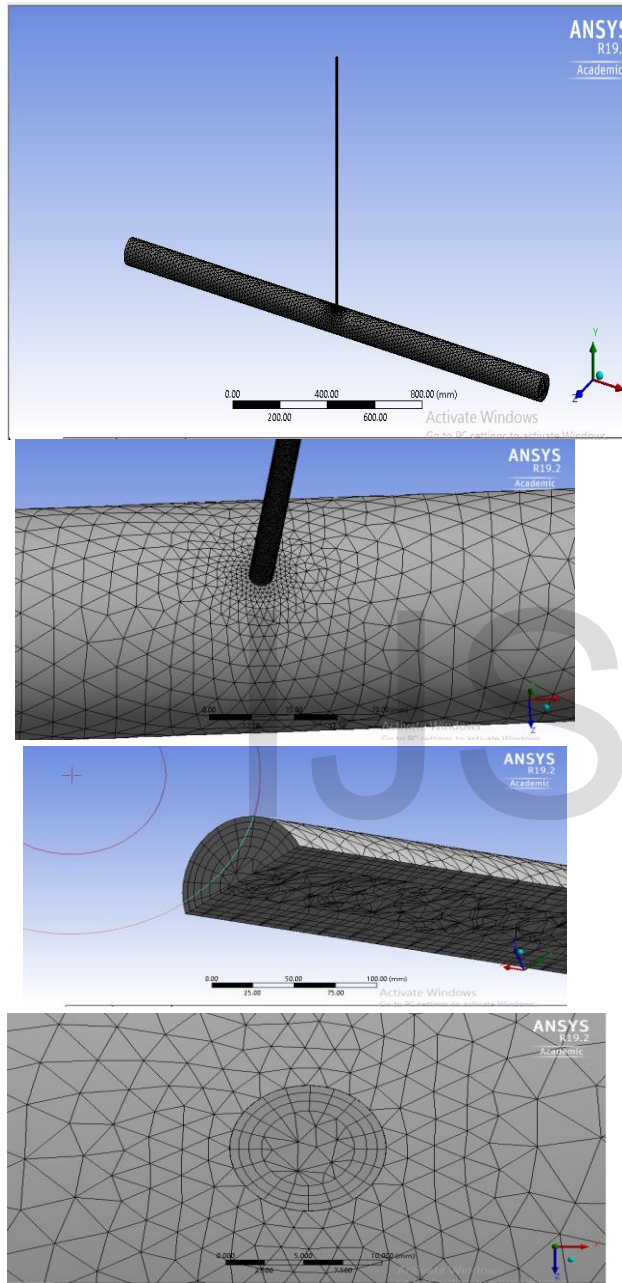


Figure 3: T-junction Meshing

3.3 Numerical model description

The geometrical model of the T-junction was created in the pre-processor IN CFD Figure 1. The diameters of adjacent branches were 10mm and the straight pipe is 100mm. The length of the straight pipe was 2000 mm and the length of the adjacent branch was

1000mm. The boundary conditions were set on the corresponding surfaces. The condition *wall* (i.e. solid surface with defined smooth surface) is used for the pipe walls.[8] The computational mesh set in this way was consequently used for the analysis of the velocity and the pressure field in software CFD the setting of the solver was chosen according to the dominant flow i.e. unsteady.

$$P_1 + \left(\frac{1}{2}\rho V_1^2\right) + \rho gh_1 = const. \text{ (Eq. 1)}$$

Static pressure = Dynamic pressure

So from calculations we have

$P_1 = 27092 Pa$, $P_2 = 42331.25 Pa$, To find the final velocity at the outlet we have;

$$\rho_1 V_1 A_1 + \rho_2 V_2 A_2 = \rho_3 V_3 A_3 \text{ (Eq. 2)}$$

$$V_3 = 2.025 \text{ m/s}$$

To find the outlet temperature we have $T_1 V_1 +$

$$T_2 V_2 = T_3 V_3 \text{ (Eq. 3)}$$

$$T_3 = 125.9 \text{ }^\circ\text{C} = 398.9 \text{ K}$$

Predicting and calculating the desired wall spacing; During the preprocessing stage we will need to know the suitable size for the first layer of grid cells (inflation layers) so that Y^+ is in the desired range. Principal parameter used to specify the pipe flow regime is the Reynolds number.

$$Re = \frac{\rho V D}{\mu} = \frac{V D}{\nu} \text{ (Eq. 4)}$$

From the simulation we have density of mercury and kinematics viscosity;

$$\eta_{Hg} = \rho_{Hg} \times \nu_{Hg} \text{ (Eq. 5)}$$

$$\rho_{Hg} = 13546 \text{ Kg/m}^3,$$

$$\eta = 0.001552 \text{ pa.s}, \nu = 1.145 \times 10^{-7}$$

Using the given parameters for Reynold number for large diameter pipe D_1 and Small diameter D_2 pipe we have ;

$$Re_1 = \frac{\rho V D_1}{\mu}, \quad Re_2 = \frac{\rho V D_2}{\mu}$$

Calculating for each diametrical pipe using the average velocity ,density and dynamic viscosity have ;

$$Re_1 = 1754385.9, \quad Re_2 = 175438.59.$$

3.3.1 Delta y calculation and dimensionless y+ calculations

Calculating wall distance for a given Y^+ ;

$$y^+ = \frac{\rho U_\tau y}{\mu} \leftrightarrow y = \frac{y^+ \mu}{U_\tau \rho} \quad (Eq. 6)$$

The target y^+ value and fluid properties are known from the ANSYS CFD simulation, so we need friction velocity U_τ to each inlet 1 and inlet 2 which is defined as;

$$U_\tau = \sqrt{\frac{\tau_w}{\rho}} \quad (Eq. 7)$$

The wall shear stress τ_w , can be found from the skin friction coefficient C_f to each inlet 1 and inlet 2, defined as ;

$$\tau_w = \frac{1}{2} C_f \rho U_\infty^2 \quad (Eq. 8)$$

$$C_f = 0.058 Re^{-0.2} \quad (Eq. 9)$$

Using the given parameter for skin friction coefficient wall shear stress and friction velocity (C_f, τ_w and U_τ) for larger diameter pipe D1 and for the small diameter pipe D2 we have ;

$$C_{f1} = 3.27 \times 10^{-3}, \quad C_{f2} = 5.18 \times 10^{-3}$$

$$\tau_{w1} = 88.5 \text{ kg/m.s}^2, \quad \tau_{w2} = 140.33 \text{ kg/m.s}^2$$

$$U_{\tau1} = 0.08 \text{ m/s}, \quad U_{\tau2} = 0.105 \text{ m/s}$$

So in order to estimate the wall distance from the above calculations we can iterate different value of Delta y until we get the better spacing (wall distance). The first calculations of Delta y is as follows

$$y_1 = \frac{y_1^+ \mu}{U_{\tau1} \rho}, \quad y_1 = 4.8 \text{ mm}$$

$$y_2 = \frac{y_2^+ \mu}{U_{\tau2} \rho}, \quad y_2 = 0.65 \text{ mm}$$

The final Delta y value for the large diameter D1 (horizontal pipe)and for the small diameter (vertical pipe)is acceptable as 4 mm for the large pipe D1 (inflation 1) and 0.55 mm for small pipe D2(inflation 2)and the y plus value is as follows .Dimension less Y^+ value small pipe is 2223.26 and for the large pipe 446.65.

Table 2: Iterations to find the Delta y value from

For	Reynold no.(Re)	Simulatio n y^+	Iteration 1 Delta y[mm]	Simulatio n y^+	Iteration 2 Delta y[mm]
Large pipe (1)	1754385.9	3512.45	4.8	2925.5	4.21
Small pipe (2)	175438.59	619.846	0.65	543	0.5528
for	Reynold no.(Re)	Simulatio n y^+	Iteration 3 Delta y[mm]	Simulatio n y^+	Iteration 4 Delta y[mm]
Large pipe (1)	1754385.9	2915.6	4.1	446.65	4
Small pipe (2)	175438.59	514.166	0.551	2223.29	0.55

simulation

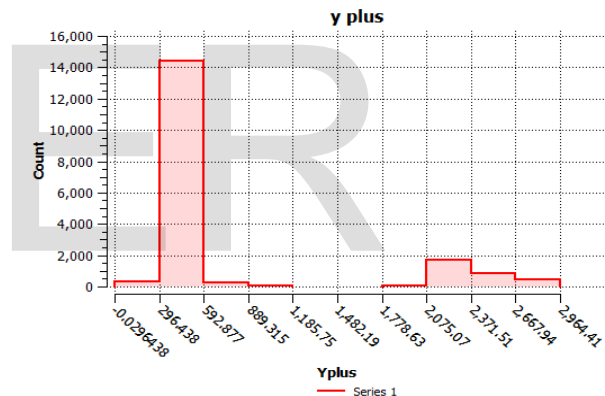


Figure 4 :Final y+ value chart from the CFD simulation

3.3.2 Velocity distribution and average velocity.

The fullness increases with the Reynolds number, and the velocity profile becomes more nearly uniform, lending support to the commonly utilized uniform velocity profile approximation for fully developed turbulent pipe flow. the flow speed at the wall of a stationary pipe is always zero (no-slip condition)[9].

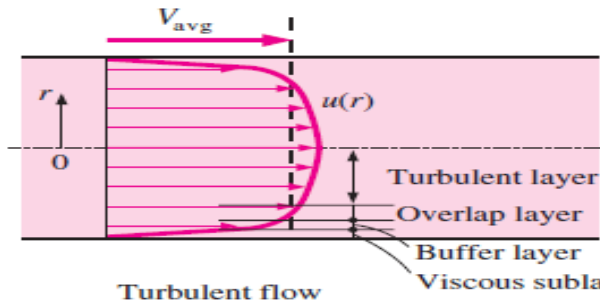


Figure 5: The velocity profile in fully developed pipe flow is parabolic in laminar flow, but much fuller in turbulent flow.[10]

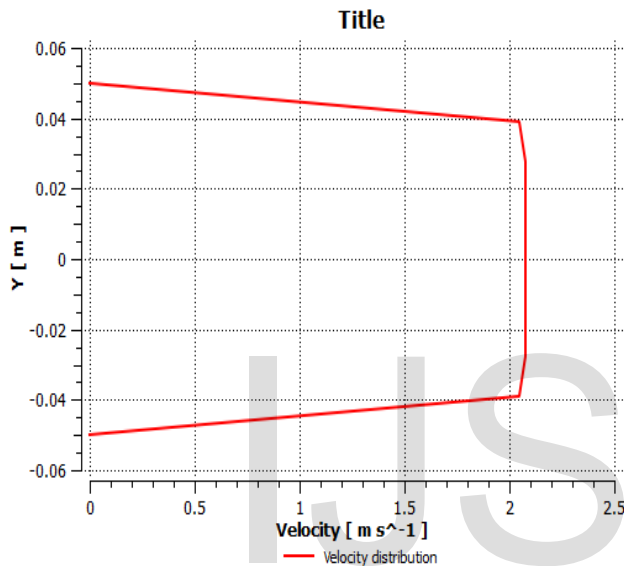


Figure 6 : velocity distribution and average velocity from CFD simulations.

3.3.3 Head loss of T junction

The frictional head loss h_f is calculated with the pressure drop Δp of straight pipe. where λ is frictional resistance coefficient, L and D are length and diameter of straight pipe respectively. Δp is the pressure drop between the inlet and outlet.

$$\lambda = \frac{\Delta p_{friction} D}{P_{Dynamics} L} \quad (Eq. 10)$$

TO find the dynamic pressure we can

$$P_{Dynamic} = \frac{\rho V^2}{2}$$

$$\Delta P_{friction} = P_{static in} - P_{static out}$$

$$P_{static out} = 0, \quad \Delta P_{friction} = P_{static in}$$

Then we have the following;

$$\lambda = \frac{p_{static} D}{P_{Dynamics} L} \quad (Eq. 11)$$

From the above equation we have λ value for the two pipe having different diameter.

$$\lambda_1 = 0.025, \quad \lambda_2 = 0.0542$$

where ρ denotes the fluid density, and g is gravitational acceleration. h_{f1} and h_{f2} are upstream frictional head loss and downstream frictional head loss, respectively, they can be written as

$$h_{f1} = \frac{\lambda_1 L V^2}{2Dg}, \quad h_{f2} = \frac{\lambda_2 L V^2}{2Dg} \quad (Eq. 12)$$

$$h_{f1} = 50.9mm, \quad h_{f2} = 1100mm$$

3.3.4 Pressure loss

Using parameters for each pipe we can find the pressure loss. Using dynamic pressure at inlet 1 inlet 2 and outlet we can calculate in each region.

$$\Delta P = P_D \lambda \times \frac{L}{D} \quad (Eq. 13)$$

Through calculations the pressure drops at inlet 1 is 6748.14 pa, for $\lambda = 0.025$ and the pressure drop at inlet 2 is 22856.1pa for $\lambda = 0.0542$, and at the outlet is 9968.6pa for $\lambda = 0.034$.

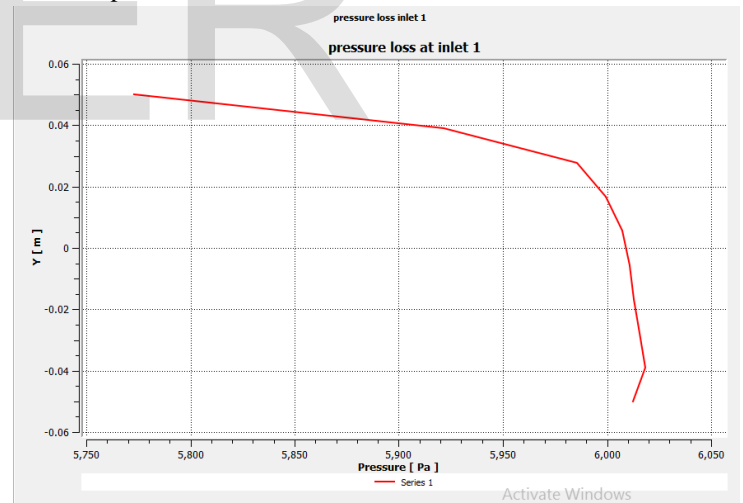


Figure 7: pressure loss at the inlet 1

3.3.5 Drag coefficient

For the t junction having two different diameters the drag coefficient between inlet 1 and outlet is as follows.

$$C_{D1} = \frac{P_{D1} - P_{D3}}{P_{D1}} \quad (Eq. 14)$$

The drag coefficient between in Inlet 2 and outlet;

$$C_{D2} = \frac{P_{D2} - P_{D3}}{P_{D2}} \quad (Eq. 15)$$

Through calculation we have;

$$C_{D1} = -0.0861613, C_{D2} = 0.304759$$

The mass flow rate at the inlet is equal to mass flow rate at outlet. From this we can identify;

$$Q_1 + Q_2 = Q_3 \quad (Eq. 16)$$

$$V_3 = \frac{V_1 A_1 + V_2 A_2}{A_3} \quad (Eq. 17)$$

3.3.6 Outlet temperature

Using the mass flow rate, we have $V_3 = 0.025 \text{ m/s}$ In order to calculate **Outlet temperature**: apply the conservation of Energy equation,

Heat Energy (In) = Heat Energy (Out),

Heat Flux (In) = Heat Flux (Out), Heat Flux 1 + Heat Flux 2 = Heat Flux 3

$$\square \text{ Heat Flux} = m \cdot C_p \times \Delta T$$

Where (m.): mass flow rate, (Cp): Specific Heat Capacity, $C_p = 2198.34 \text{ [J / kg. K]}$. Or we can calculate using the following formula.

$$T_3 = \frac{V_1 T_1 + V_2 T_2}{V_3} \quad (Eq. 18)$$

After calculating using the given parameters we have $T_3 = 398.825 \text{ K}$.

4. CFD simulation and result

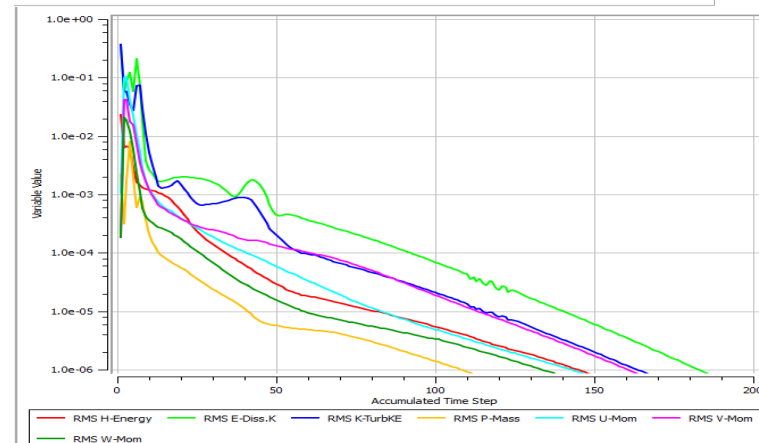
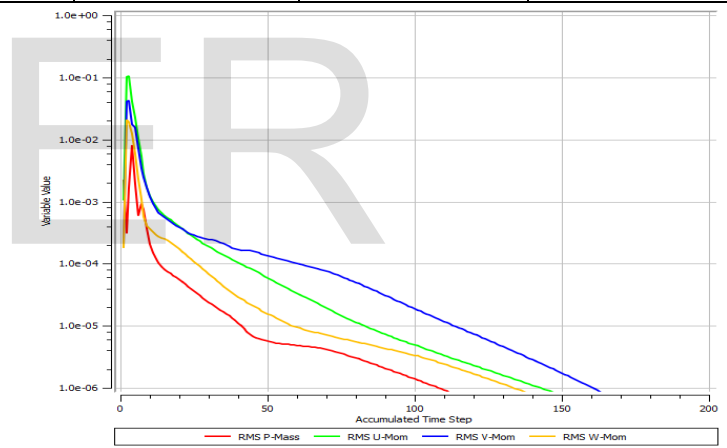
In this section, calculation and results demonstrated with and without simulation software, and the error between the two methods. When the inlet velocity v_1 is equal to 2m/s, partial streamlines are shown in Fig. (8). It shows that the branch with the inlet velocity of 2.5 m/s is on the upper part of flow channel due to the jacking role of the mainstream, and it flows to the pipe center gradually during the mixing of two streams. However, the branch with the inlet velocity of 2.5m/s forces the mainstream to significantly turn to the opposite side of junction after it flows into the junction, and a vortex is formed at the downstream of junction as a result Fig. (9). When the velocity of branch is higher, junction of tee has larger effect on the surrounding pressure, which is characterized by the increasing of pressure in the upstream and decreasing of pressure in the downstream of junction.

4.1 Residual

Results from ANSYS CFX Solver As shown in chart below, the residuals of model in final simulation are below 10^{-5} .

Table 3. Results with CFD simulation, Theoretical calculations and the error.

	Simulation		
	Velocity [m s ⁻¹]	Temp. [K]	Pressure [Pa]
Inlet 1	1.996e+00	3.631e+02	2.699e+04
Inlet 2	2.495e+00	3.031e+02	4.217e+04
Outlet	2.019e+00	3.624e+02	2.932e+04
Theoretical			
Inlet 1	2.000e+00	3.630e+02	2.709e+04
Inlet 2	2.500e+00	3.030e+02	4.217e+04
Outlet	2.025e+00	3.989e+02	2.761e+04
Error			
Inlet 1	-1.842e-01	4.132e-02	-3.671e-01
Inlet 2	-1.912e-01	4.950e-02	1.075e-03
Outlet	-3.024e-01	-9.155e+00	-6.207e+00



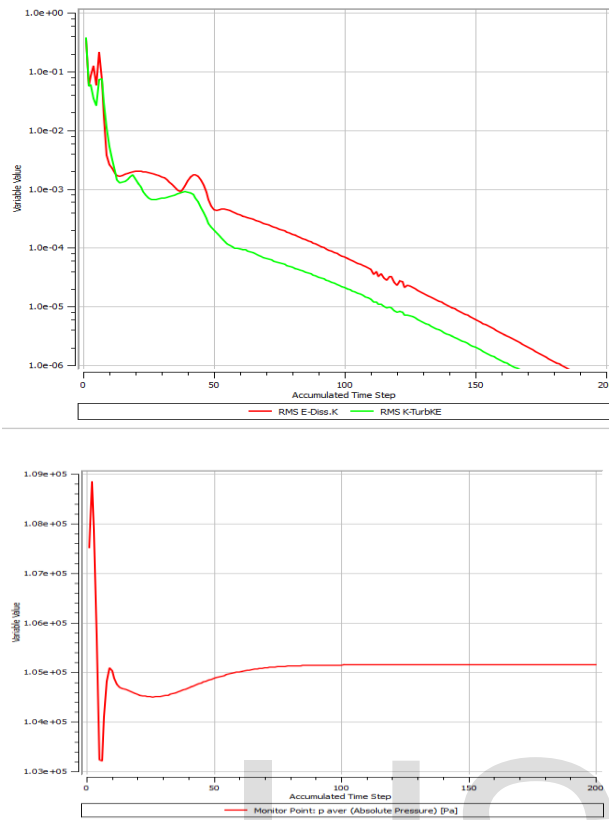


Figure 8: Final residual convergence of the hall simulation after iterations.

4.2. Simulation Equations using ANSYS and result

EXPRESSIONS:

Accumulated Time Step = 218
 Current Time Step = 218
 Drag coefficeint = $((\text{pressure inlet 1 sim} - \text{pressure outlet simu}) / \text{pressure inlet 1 sim})$
 Dynamic pressure = $\text{areaAve}(\text{Density})@inlet 2 * 2$
 Dynamic viscocisty ave = $\text{volumeAve}(\text{Dynamic Viscosity})@Default Domain$
 Lamda = 0.025
 Pressure error inlet 2 = $((\text{pressure inlet 2 simu} - \text{pressure inlet 2 th}) / \text{pressure inlet 2 th}) * 100$
 Pressure error outlet = $((\text{pressure outlet simu} - \text{pressure outlet th}) / \text{pressure outlet th}) * 100$
 Pressure loss inlet 1 = $((\text{areaAve}(\text{pressure inlet 1 sim})@inlet 1 * \text{Lamda}) / 0.1)$
 Reference Pressure = 1 [atm]
 Renould nu = $(\text{volumeAve}(\text{Density})@Default Domain * \text{Velocity inlet 1 sim} * 0.1) / (\text{volumeAve}(\text{Dynamic Viscosity})@Default Domain)$
 Sequence Step = 218

Temprature error inlet 1 = $((\text{Temprature inlet 1 sim} - \text{Temprature inlet 1 th}) / \text{Temprature inlet 1 th}) * 100$
 Temprature error inlet 2 = $((\text{Temprature inlet 2 sim} - \text{Temprature inlet 2 th}) / \text{Temprature inlet 2 th}) * 100$
 Temprature error outlet = $((\text{Temprature outlet sim} - \text{Temprature outlet th}) / \text{Temprature outlet th}) * 100$
 Temprature inlet 1 sim = $\text{areaAve}(\text{Temperature})@inlet 1$
 Temprature inlet 1 th = 363 [K]
 Temprature inlet 2 sim = $\text{areaAve}(\text{Temperature})@inlet 2$
 Temprature inlet 2 th = 303 [K]
 Temprature outlet sim = $\text{areaAve}(\text{Temperature})@outlet$
 Temprature outlet th = 398.926 [K]
 Time = 0 [s]
 Velocity inlet 1 sim = $\text{areaAve}(\text{Velocity})@inlet 1$
 Velocity inlet 2 sim = $\text{areaAve}(\text{Velocity})@inlet 2$
 Velocity outlet sim = $\text{areaAve}(\text{Velocity})@outlet$
 atstep = Accumulated Time Step
 ctstep = Current Time Step
 density ave = $\text{volumeAve}(\text{Density})@Default Domain$
 drag coefficient 2 = $((\text{pressure inlet 2 simu} - \text{pressure outlet simu}) / \text{pressure inlet 2 simu})$
 eddy visco = $\text{Dynamic viscocisty ave} / \text{density ave}$
 friction coefficient lamda = $(\text{massFlowAve}(\text{Pressure})@inlet 1 * 0.1[\text{m}]) / (\text{massFlowAve}(\text{Pressure})@inlet 1 * 1[\text{m}])$
 lamda 2 = 0.0542
 pressure inlet 2 th = $((\text{areaAve}(\text{Velocity})@inlet 2)^2 * \text{volumeAve}(\text{Density})@Default Domain) / 2$
 pressure error inlet 1 = $((\text{pressure inlet 1 sim} - \text{pressure inlet 1 th}) / \text{pressure inlet 1 th}) * 100$
 pressure inlet 1 sim = $\text{massFlowAve}(\text{Total Pressure-Pressure})@inlet 1$
 pressure inlet 1 th = $((\text{velocity inlet 1 th}^2) * \text{density ave}) / 2$
 pressure inlet 2 simu = $\text{massFlowAve}(\text{Total Pressure-Pressure})@inlet 2$
 pressure loss at out let = $(\text{areaAve}(\text{pressure outlet simu})@outlet * 0.034) / 0.1$
 pressure loss inlet 2 = $((\text{areaAve}(\text{pressure inlet 2 simu})@inlet 2 * \text{lamda 2}) / 0.01)$
 pressure outlet simu = $\text{massFlowAve}(\text{Total Pressure-Pressure})@outlet$
 pressure outlet th = $((\text{areaAve}(\text{Velocity})@outlet)^2 * \text{volumeAve}(\text{Density})@Default Domain) / 2$
 sstep = Sequence Step
 t = Time
 velocity error inlet 1 = $((\text{Velocity inlet 1 sim} - \text{velocity inlet 1 th}) / \text{velocity inlet 1 th}) * 100$
 velocity error inlet 2 = $((\text{Velocity inlet 2 sim} - \text{velocity inlet 2 th}) / \text{velocity inlet 2 th}) * 100$
 velocity error outlet = $((\text{Velocity outlet sim} - \text{velocity outlet th}) / \text{velocity outlet th}) * 100$
 velocity inlet 1 th = 2 [m s⁻¹]

velocity inlet 2 $th = 2.5[m\ s^{-1}]$
 velocity outlet $th = ((velocity\ inlet\ 1\ th * area@inlet\ 1) + (velocity\ inlet\ 2\ th * area@inlet\ 2)) / area@outlet$

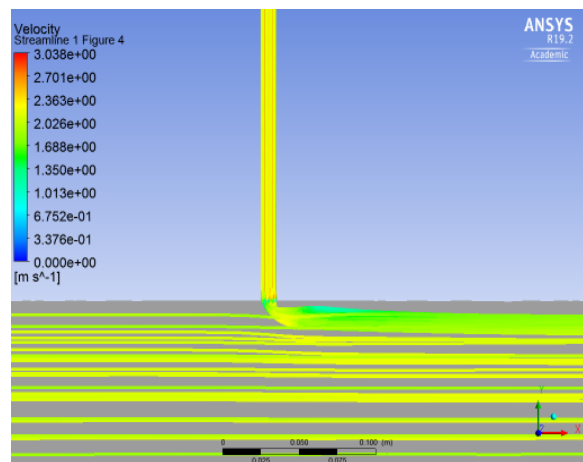


Figure 9: Velocity stream line

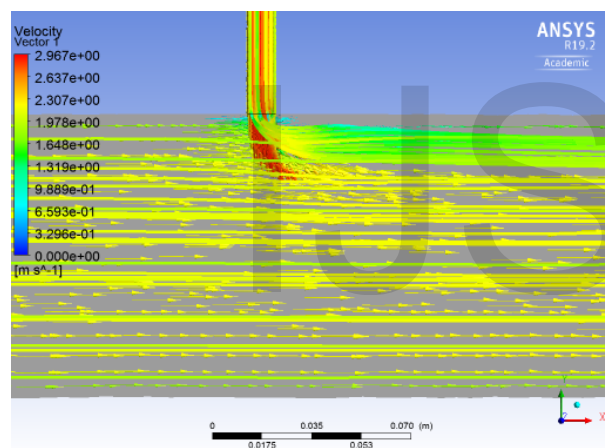


Figure 10 :Velocity vector

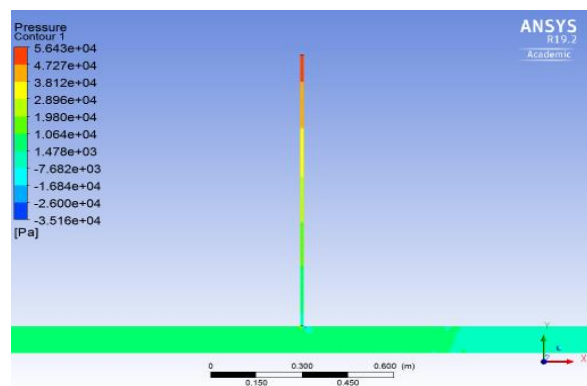


Figure 11 : Pressure counter

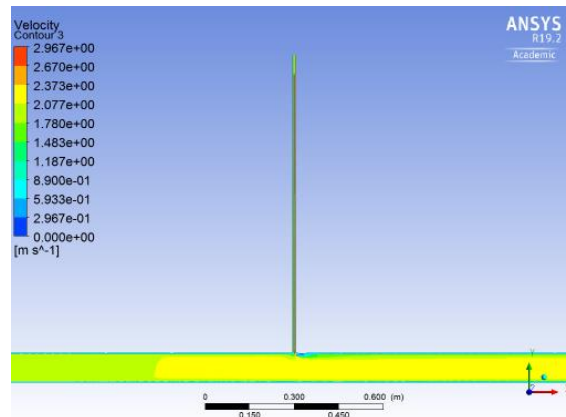


Figure 12: Velocity counter

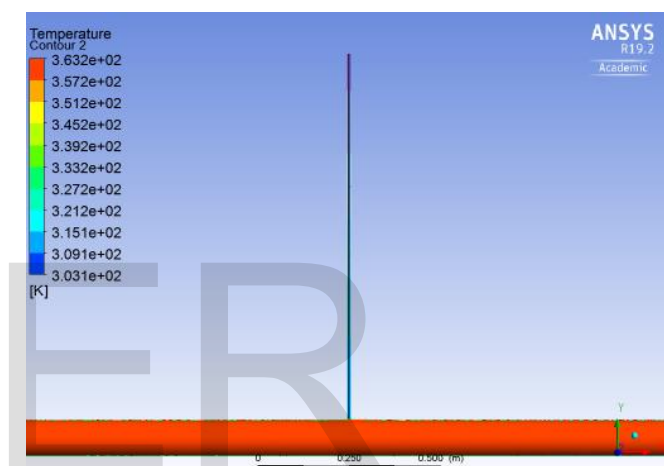


Figure 13:Temperature counter

5.Reference

[1]. M. Kotek, V. Kopecký, D. Jašíková, Recent Researches in Mechanics, Proc. of the 2nd International Conference on Fluid Mechanics and Heat and Mass Transfer 2011 (FLUIDSHEAT11) WSEAS Cofru, 169-172, (2011)

[2]. Štigler J., Šperka O., Klas R.:Mathematical model of the fluid flow in the Tee-Junction. The comparison of the CFD computation and the measurements. In 17th international conference Engineering Mechanics ,(2011),

[3]. Svatka Czech Republic, May2011, pp. 607 – 610, ISBN 978-80-87012-33-8. Miller D. S. *Internal Flow. A Guide to Losses in Pipe and Duct Systems.* The British Hydromechanics Research Association, Cranfiled, Bedford, England, (1971), pp. 329.

[4]. Maia, R. J., 1992, "Numerical and experimental investigations of the local losses in piping systems. Methods and Techniques for its systematic investigation. The specific case of the flow in a 90Tee junction" PhD thesis, University of Porto, Portugal (in Portuguese).

[5]. Ito, H. and Imai, K., "Energy Loss at 90° Pipe Junctions." Proc. ASCE, Journal of Hydraulics Division, September 1973, pp. 1353-1968.

[6] Blaisdell, F. W. and Manson, P. W. (1967), "Energy Loss at Pipe Junctions," Proc. ASCE, Journal of the Irrigation and Drainage Division, September, 93 (IR3), pp. 59-78.

[7]. Vogel, G., (1929) "Experiments to Determine the Loss at Right Angle Pipe Tees," Hydraulic laboratory practice (ed. Freeman, J.R.), pp. 470-472, Published by American Society of Mechanical Engineers.

[8]. Ze Su, (1996) "Pressure Drop in Perforated Pipes for Horizontal Wells". Ph. D., Norwegian.

[9]. Blaisdell, F. W. and Manson, P. W. (1963), "Loss of Energy at sharp -Edged Pipe Junctions in Water Conveyance systems", Agricultural Research Service, U.S. Department of Agricultural and Minnesota Agricultural Experiment Station, St. Anthony Falls Hydraulic Lab., U of Minnesota.

[10]. Štigler J 2007 Mathematical Model of the Unsteady Fluid Flow Through Tee-Junction 2nd IAHR Int. Meeting of the Work Group on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems. Scientific Buletin of the "Politehnica" University of Timisoara (Romania Transactions on Mechanics,ISSN 1224-6077) **52(6)** 83-92

IJSER