STUDY OF AIR -WATER FLOW INSIDE A 3-D CROSS JUNCTION

Ass.Lec. Dhafer Abdulameer Hamzah
University of Al-Qadisiya -College of Engineering- Mechanical Engineering

ABSTRACT

This study examines the flow of water and air in a cross junction by fluent under Ansys 12. Initially the problem will solve by using the less computationally intensive mixture model, and then turn to the more accurate Eulerian model. In this study which is using flow of water and air inside closed and vertical conduit as cross junction so the flow is water and air as mixture in inlet of pipe and the mixture leaves the pipe in three outlets as shown in **Figure (1)**, also the phase of the air is assume to be bubble flow, so the diameter of the bubble of air will be present. The slip velocity equation need to solve since there will be significant difference in velocities for the different phases. The study used solid work programme to work the figure and then export it to fluent of Ansys. The study used the mixture model and Eulerian model to investigate the distribution of air inside the cross pipe and the where of the effect of static pressure inside the cross conduit for design. Also the contour of velocity magnitude is used to know the moving and distribution of the flow. The comparison between the two model are happen to know the range of converge for results for two model.

Keywords: bubble flow, mixture model, Eulerian model, multiphase flow, volume fraction

الخلاصة

هذا البحث اختبر جريان الماء والهواء في توصيلة متعددة. في البداية سوف تحل المسالة بواسطة موديل المزيج الأقل حسابات ومن ثم استخدام موديل اؤيلر الأكثر دقة في هذه الدراسة والتي استخدمت جريان الماء والهواء في داخل قناة مغلقة عمودية كتوصيلة متقاطعة لذا فان الجريان المستخدم هو الماء والهواء كخليط في مدخل الأنبوب حيث يغادر الخليط الأنبوب في ثلاثة مخارج كذلك الطور للهواء سوف يفرض بان يكون على شكل فقاعات لذا فان قطر الفقاعة سوف يتم استخدامه. سرع الانزلاق سوف تحل في موديل المزيج بسبب وجود اختلاف في السرعة للطورين. الدراسة استخدمت برنامج solid work لغرض رسم الشكل ومن ثم أرسالة إلى برنامج Ansys . الدراسة استخدمت موديل المزيج وموديل اؤيلر لتخمين توزيع الهواء في داخل الأنبوب المتقاطع وكلك معرفة مكان تأثير الضغط الساكن في داخل القناة المتقاطعة لغرض التصميم. كلك تم رسم مخطط كمية السرعة لعرفة حركة وتوزيع الجريان. المقارنة بين الطريقتين قد تمت لغرض معرفة مدى قرب النتائج لكلا الطريقتين.

1.INTRODUCTION

The conventional term for the concurrent flow of air and water is two-phase flow. Here, phase refers to one of the states of matter (gas, liquid, or solid). Technically the term two phase flow should be reserved to describe the motion of a substance which is present in two of its phases, such as a flow of ice and water. The word multi component is a better description of flows which do not consist of the same chemical substance, such as air and water. If both components move in the same direction, the flow is termed concurrent flow.

If the components move in opposite directions, the flow is countercurrent. Closed conduit flow can be classified according to the type of pattern that develops. The flow patterns which develop depend upon the air flow rate relative to the water-flow rate and the slope of the conduit. For example, the flow patterns in horizontal conduits have been defined by (**Baker 1954** and **Kai Yan et al 2009**), bubble flow is one of this pattern that the water as spherical or spherical cap bubbles which are small with respect to the conduit diameter. The bubbly flow pattern is characterized by a suspension of discrete bubbles in a continuous liquid. There are numerous regimes of bubbly flow. Void fraction range from the extreme case of a single isolated bubble in a large container to the quasi-continuum flow of a foam, containing less than 1 percent of liquid by volume. Interaction between the forces that are due to surface tension, viscosity, inertia, and buoyancy produces a variety of effects which are quite often evidenced by different bubble shapes and trajectories (**Wallis 1969**). Engineering applications include bubble columns for promoting mass transfer, high pressure evaporators, flash distillation, fire fighting foams, pumping of beer, champagne,, and ice cream, cryogenics, foam of impurities, sewers to handle the efflux from washing machines, instant lather for shaving, under water breathing and control of wave action in harbors.

2.PROBLEM DISCRIPTION

This problem considers an air-water mixture flowing upwards in a duct and then splitting in a cross junction. The ducts are 25 mm in width, the inlet section of the duct is 125 mm long, and the top and the side ducts are 250 mm long. The schematic of the problem is shown in **Figure 1**. The flow rate of air and water are enter the cross junction in y-direction and splitting in three direction x, y, z.

3. VOLUME FRACTION CALCULATION

Every part of the flow field is occupied by one or other component so the volume fraction represents the fraction of an element of volume which is occupied at any instant by component 2. However for most purpose a volume much larger than the discrete particles (drops or bubbles) and

(B) then represents an average volumetric concentration. Usually B is measured as an average over the whole flow cross section and a sufficient length of duct to eliminate local fluctuations. Thus if a pipe is suddenly isolated by closing valves at both ends, the contents can be analyzed and the total volume of component 2 can be determined (**CCrowe 1998**). The below equations define the volume fraction for air:

$$B=V_2/AL \tag{1}$$

$$B=Q_2/Q_1+Q_2 \tag{2}$$

4. VELOCITY CALCULATION

The volume flux and volume fraction for two phase are used for estimate the velocity for both phases as the equations below (**Taylor 1962**):

$$J_1=Q_1/A j_2=Q_2/A \tag{3}$$

$$v_1=j_1/1-B, v_2=j_2/B \tag{4}$$

5. BUBBLE FORMATION

It is only rarely that bubbly flow is the final stable equilibrium flow regime in a given duct gas bubbles suspended in fluids usually tend to agglomerate and lose their identity, and when evaporation or condensation occur the existence of small bubbles is only transitory. For these reasons it is particularly important to understand the ways in which the bubbles can be formed. Moreover, the bubble size has influence on the dynamics of the bubble mixture and must be specified in term of the mechanism of bubble generation (**Kutateladeze et al 1958** and **Brennen 1998**) the following equation can evaluate the radius of the bubble in term of radius of the orifice

$$R_b=(\sigma R_o/g(\rho_f-\rho_g))^{1/3} \quad (5)$$

6. THORY AND TRANSPORT EQUATION FOR THE STANDARD k-ε MODEL

The numerical simulation of fluid flow is achieved by solving compressible, viscous (standard k-ε model)with three dimensional. In general, basic equations of fluid flow are known, therefore in the current paper we emphasized on the solution of the three-dimensional turbulent fluid flow and heat transfer in a cross junction. The standard k-ε model is a semi-empirical model based on model transport equations for the turbulent kinetic energy (k) and its dissipation rate (ε) (**Anderson,1990**) *k* and ε are obtained from the following semi empirical transport equations(6.7), as equation (1&2) (**Choudhury,1993)** and (**ANSYS Inc. 2009**):

$$\left.\begin{aligned} div\left(\rho KV\right) &= div\left\{\left(\mu_t/\sigma_k\right) grad k\right\} \\ &+ 2\mu_t E_{ij} E_{ij} - \rho\varepsilon \end{aligned}\right\} \quad (6)$$

$$\left.\begin{aligned} div\left(\rho\varepsilon V\right) &= div\left\{\left(\mu_t/\sigma_\varepsilon\right) grad\varepsilon\right\} + \\ &C_{1\varepsilon}\frac{\varepsilon}{k}2\mu_t E_{ij} E_{ij} + C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \end{aligned}\right\} \quad (7)$$

7. RESUELTS AND DISSCTION

The mixture turbulence model is applicable when phases separate, for stratified (or nearly stratified) multiphase flows, and when the density ratio between phases is close to 1. In these cases, using mixture properties and mixture velocities is sufficient to capture important features of the turbulent flow (**ANSYS, Inc. 2009**). Also due to the Eulerian model solves individual momentum equations for each phase, you can choose the phase for which solution data is plotted.

ITERATIONS AND RESIDUALS:

There are three indicators that convergence has been reached in **Figure 2** for two model (mixture model & Eulerian model) for present Reynolds number:

• For mixture model the residuals have decreased to a sufficient degree. The solution has converged when the convergence criterion for each variable has been reached. The default criterion is that each residual will be reduced to a value of less than $1e^{-04}$, except the volume fraction for air which is be which the default criterion is $1e^{-02}$. Also for the Eulerian model which is start at 400 iteration and ends at 775 iteration the limits for all residuals are same thing in mixture model.

• The solution no longer changes with more iterations. Sometimes the residuals may not fall below the convergence criterion set in the case setup. However, monitoring the representative flow variables through iterations may show that the residuals have stagnated and do not change with further iterations. This could also be considered as convergence.

• The overall mass, momentum, energy, and scalar balances are obtained for example the **table 1** define that the net of mass for all outlets and inlet is equal zero.

• The mixture model is a simplification of the Eulerian model and is valid only when bubble velocity is in the same direction as water velocity. This assumption can be violated in the recirculation pattern. The Eulerian model is expected to make a more realistic prediction in this case. The study used the solution obtained using the mixture model as an initial condition for the calculation using the Eulerian model.

PRESSURE DISTRIBUTION

For the two model in the **Figure 3** the distribution of the static pressure in the cross junction is similar, wherever the static pressure in x and z directions have the same values in both models and its value is the mean value for maximum and minimum in the inlet and y-direction outlet respectively because of the effect of the gravity and the air phase which is found as bubbles.

VELOCITY DISTRIBUTION

In the three dimension cases we note that the contours of the velocity magnitude in this case **Figure 4**, for two model is not clear enough to observe the all varies of the velocity distribution but in spite of that we use the contours and without the effect of the wall of the junction we can observe that the distribution of the velocity is same in two dimension wherever the maximum velocity in the inlet and mean velocity in the all outlets so the velocity of the water is be the most effect on the all parts of the junction.

VOLUME FRACTION OF THE AIR

When gravity acts downwards, it induces stratification in the side arms of the cross junction. In **Figure 5**, you can see that the gas (air) tends to concentrate on the upper part of the side arm. In this case, gravity acts against inertia that tends to concentrate gas on the low pressure side, thereby creating gas pockets. In the vertical arm, the gas travels upward faster than the water due to the effect of gravity, and therefore there is less separation. The outflow split modifies the relation between inertia forces and gravity to a large extent, and has an important role in flow distribution and on the gas concentration.

CONCOLUSION

In our study the results refer to some points that must take in our future work which are:

1. ANSYS FLUENT, it is good practice to use your first-order solution as a starting guess for a calculation that uses a higher-order discretization scheme and, optionally, an adapted mesh.
2. The solution obtained with the mixture model was used as a starting point for the calculation with the Eulerian model. After completing calculations for each model, you displayed the results to allow for a comparison of the two approaches.

3.Since the Eulerian model solves individual momentum equations for each phase, we can choose the phase for which solution data is plotted

References

Anderson, John D., Jr., "Modern Compressible Flow: with Historical perspective", 2nd ed. McGraw-Hill, New York, 1990.1.
ANSYS, Inc. March 12, 2009.," Using the Mixture and Eulerian Multiphase Models"
Baker,O., Oil gas journal, vol. 53, no. 12,pp. 185-190, July 26, 1954.
C. Crowe, M. Sommerfield, and Yutaka Tsuji. Multiphase Flows with Droplets and Particles. CRC Press, 1998.
C.E. Brennen. Cavitation and Bubble Dynamics. Oxford University Press, 1995.
Choudhury D. " Introduction to the Renormalization Group Method and Turbulence Modeling". Fluent Inc. Technical memorandum TM-107, 1993.
Kutateladze,S. S., and M. A. Styrikovich: " Hydraulic of gas liquid system" Moscow, Wright Field trans. F-TS-9814/ V, 1958.
Taylor, G. I.: Proc. Roy. Soc. (London), Vol. A210, P. 192, 1950.
Wallis,G. B.: paper no. 38, Intern. Heat transfer conf., Boulder, colo., Asme, 1961.Kai Yan, Defu Che "A coupled model for simulation of the gas–liquid two-phase flow with complex flow patterns". International Journal of Multiphase Flow.

Table 1. Flux report

Part	Mass flow rate of air (Eulerian model (kg/s)
Inlet y	1.7992739e-05
Outlet x	-8.3397726e-06
Outlet y	-1.4863522e-06
Outlet z	-8.1654916e-06
Net of mass flow rate	2.5624888e-07

Part	Mass flow rate of air (mixture model) (kg/s)
Inlet y	1.7992739e-05
Outlet x	-8.0968694e-06
Outlet y	-1.4863522e-06
Outlet z	-7.938489e-06
Net of mass flow rate	4.7102867e-07

Part	Mass flow rate of water (mixture model) (kg/s)
Inlet y	0.68698341
Outlet x	-0.26001221
Outlet y	-0.16718651
Outlet z	-0.26016954
Net of mass flow rate	-0.00038485229

Part	Mass flow rate of water (Eulerian model) (kg/s)
Inlet y	0.68698341
Outlet x	-0.25982958
Outlet y	-0.16739155
Outlet z	-0.25997168
Net of mass flow rate	-0.00020940602

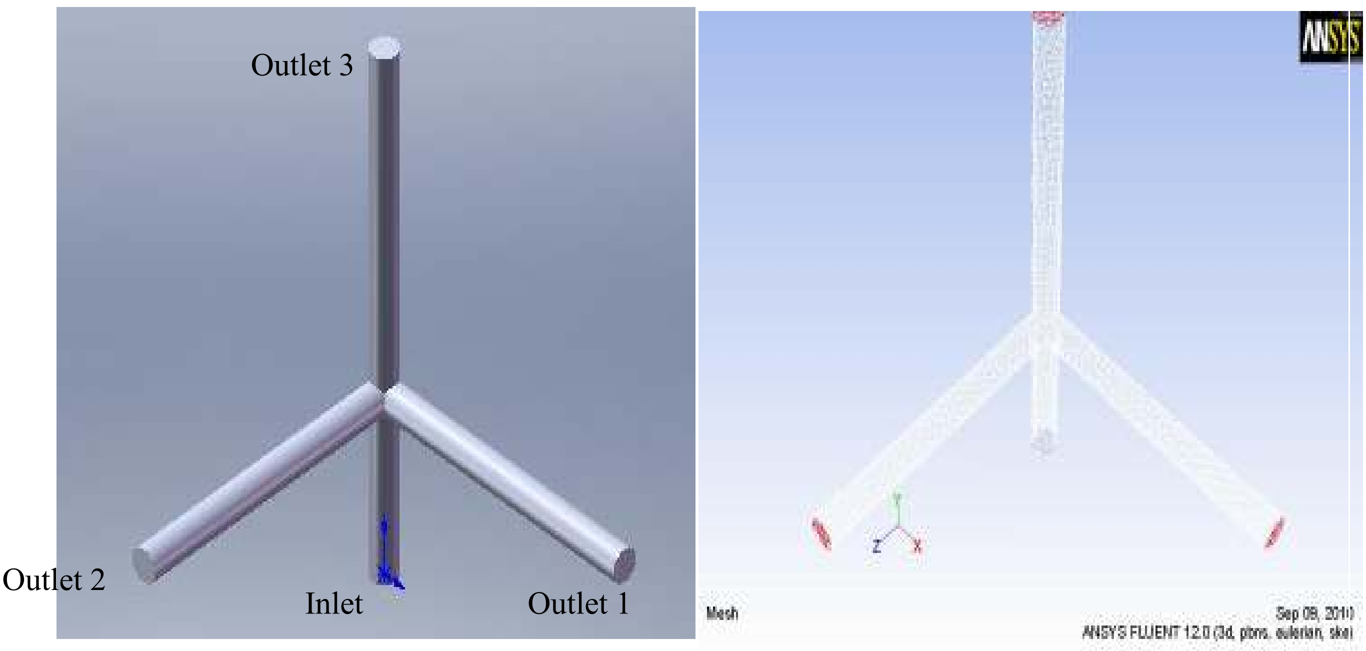


Figure 1. The geometry and mesh generation of the cross junction

Residuals of Mixture model

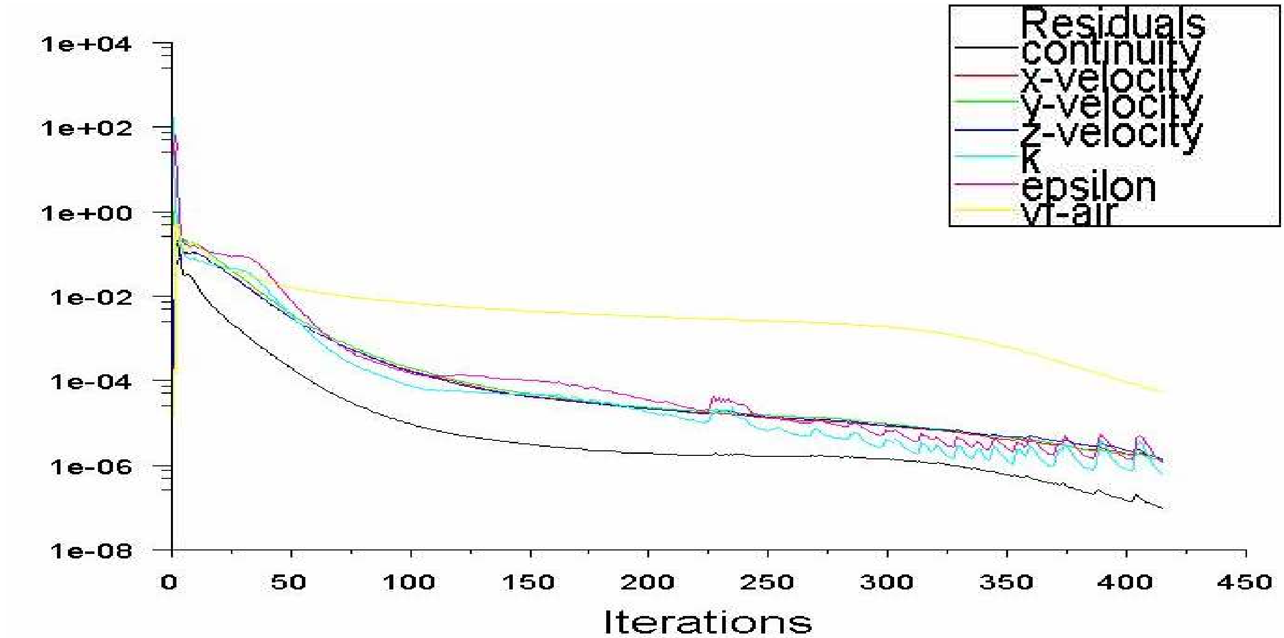


Figure 2a. Converge history for scaled residuals (using mixture model)

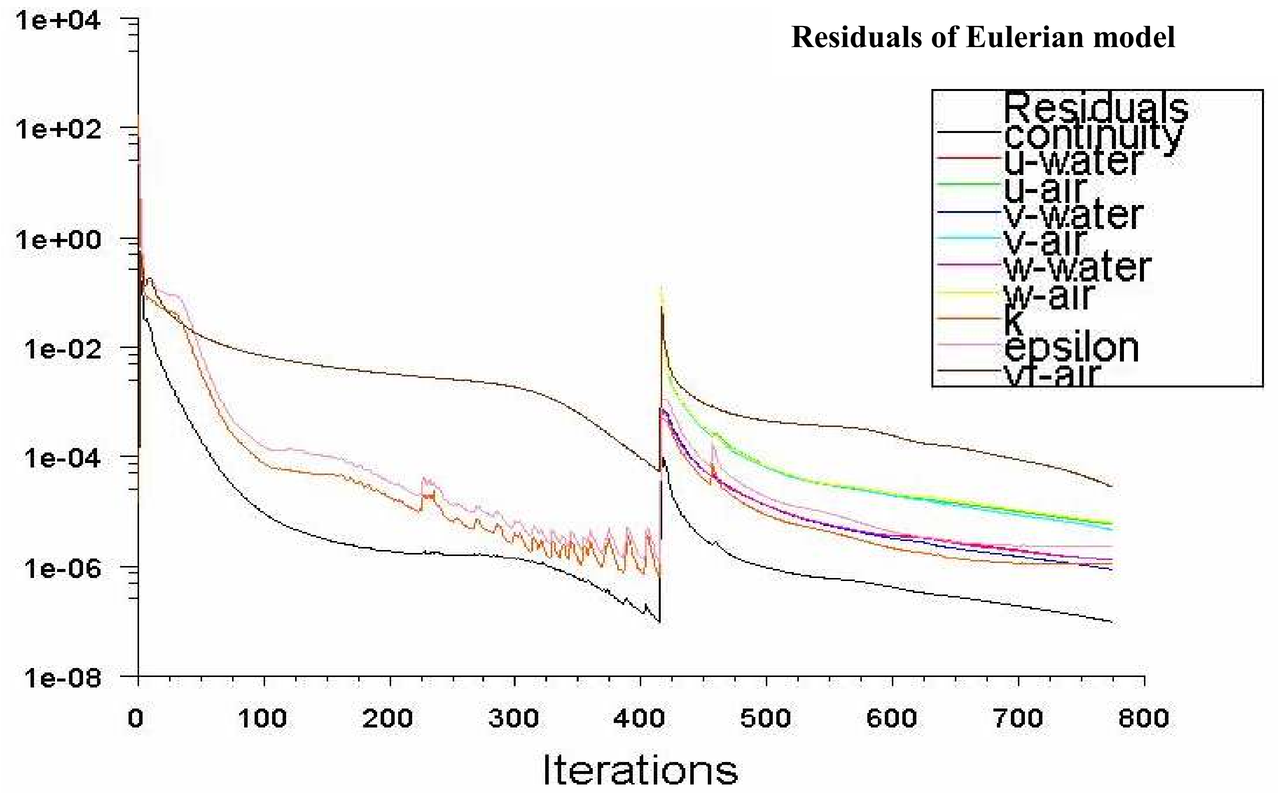


Figure 2b. Converge history for scaled residuals (After using Eulerain model)

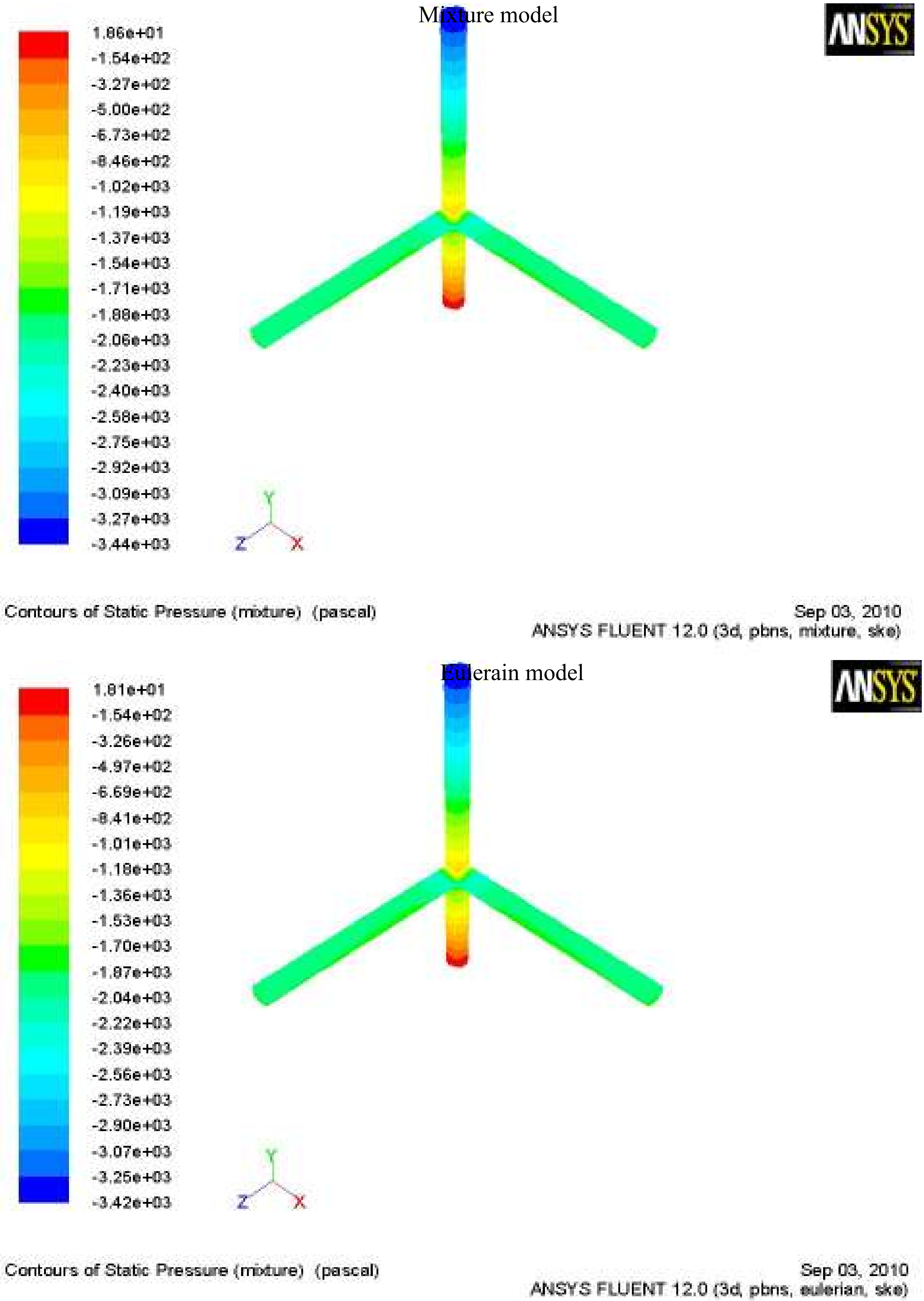


Figure 3. Contours of static pressure for mixture by mixture model and Eulerain model

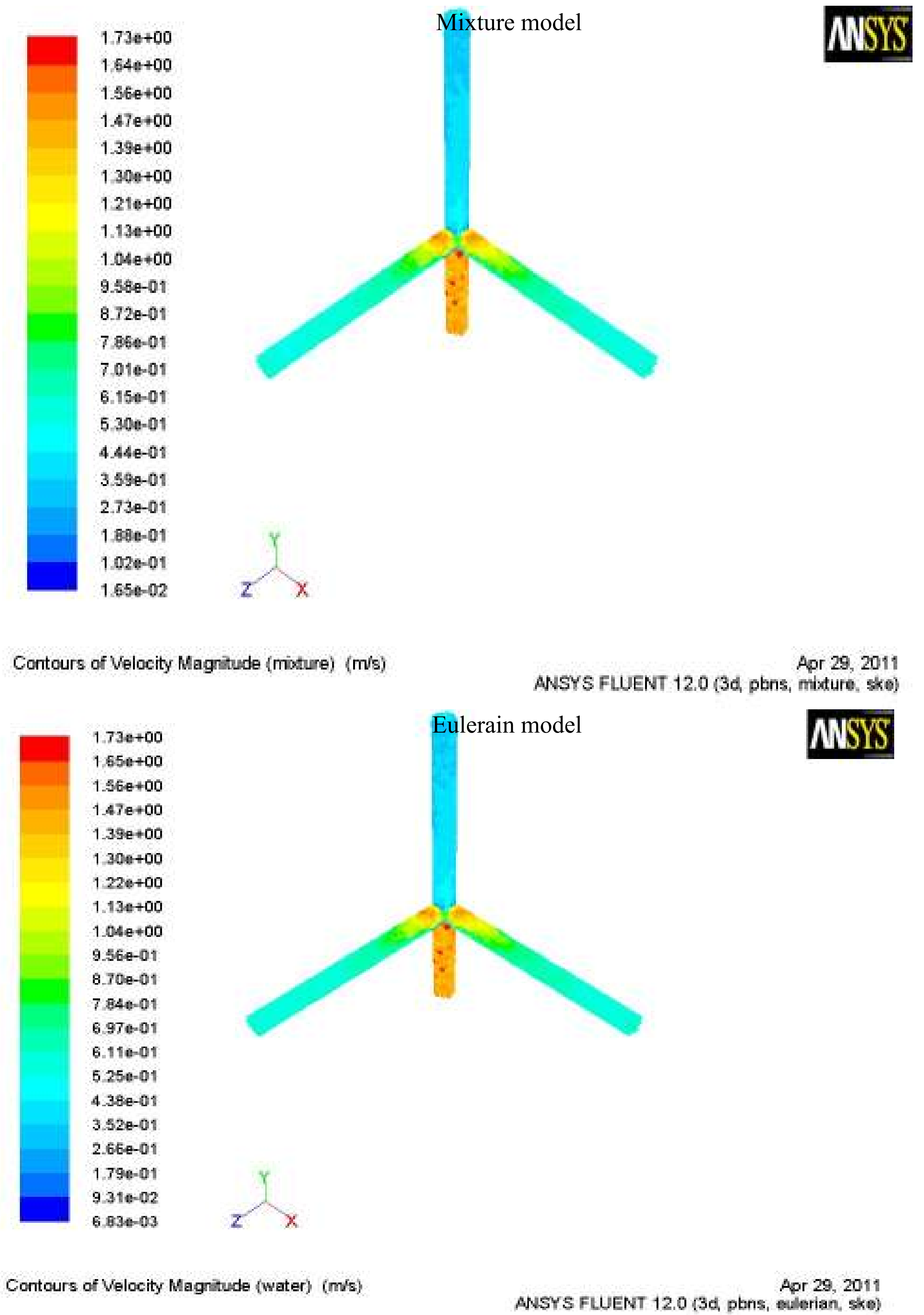
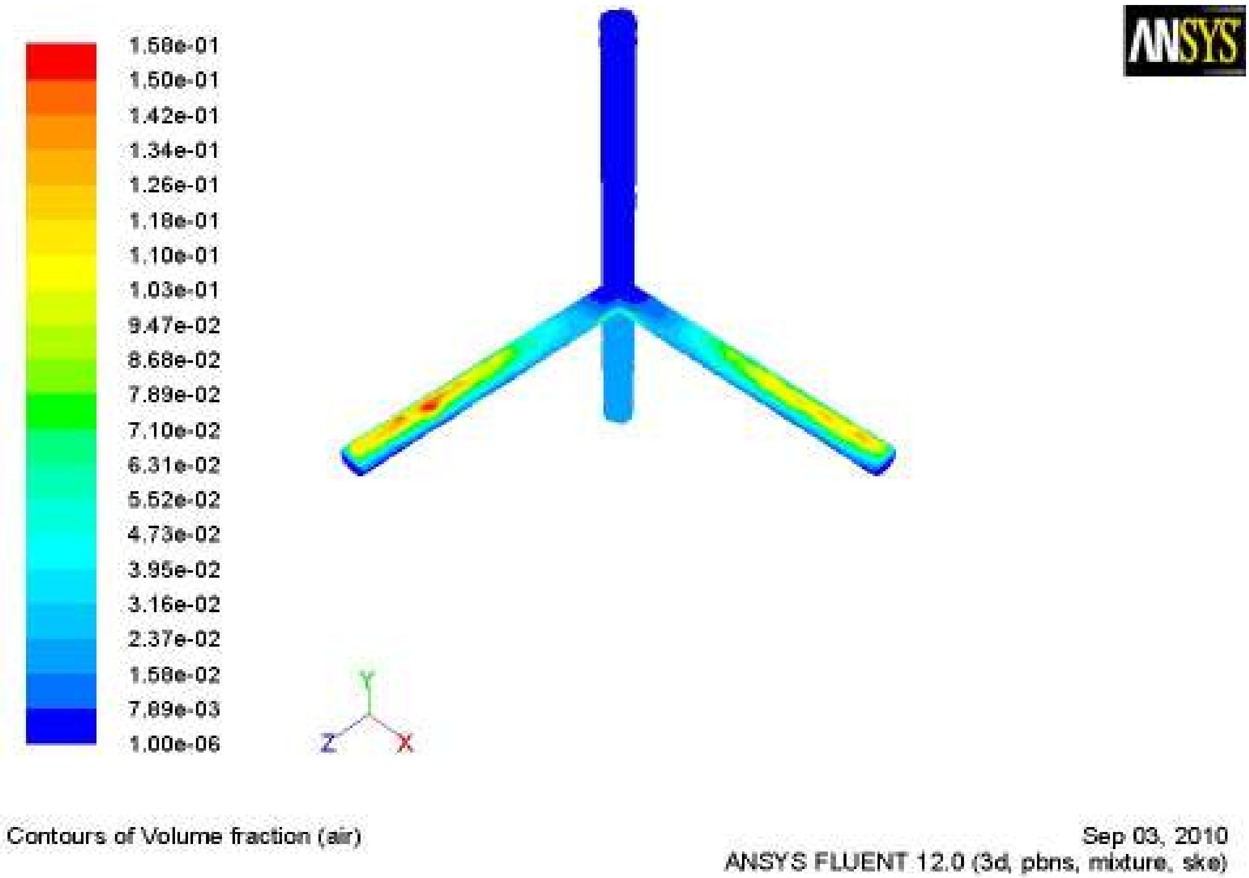


Figure 4. Contours of velocity magnitude



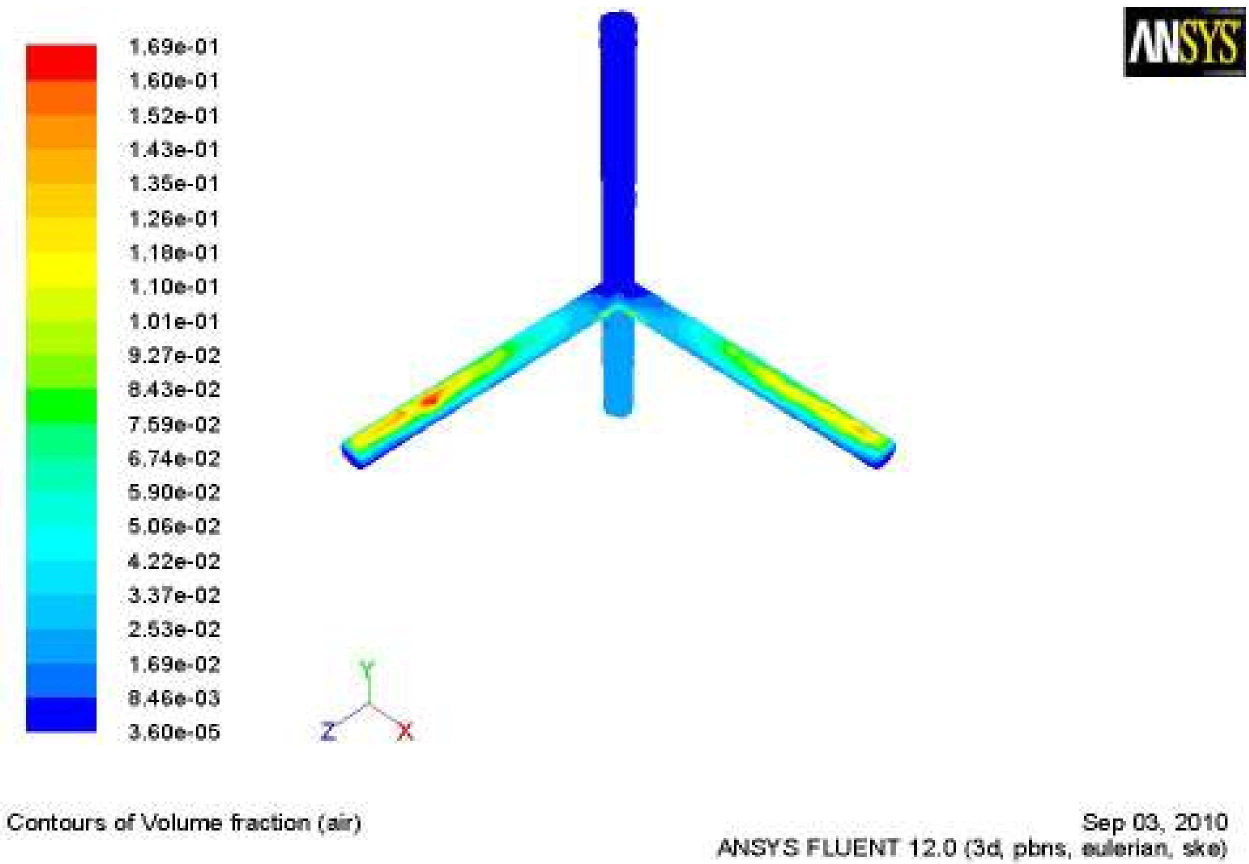


Figure 5. Contours of volume fraction

NOMENCLATURE

Symbols	Meaning	Units	Symbols	Meaning	units
C_μ, $C_{1\varepsilon}$, and $C_{2\varepsilon}$	Ansys default Constants(0.09,1.44,1.92)	-	ρ	Density	kg/m^3
C_p	Specific heat	kJ/kg.K	μ	Dynamic Viscosity	Pa.s
Eij	Edge length	m	μ_t	Turbulent Viscosity	Pa.s
I	Turbulent intensity	-	ε	Turbulent Dissipation rate	m^2/s^3
V	Mean velocity	m/s	ν	Kinematics Viscosity	m^2/s
A	Cross section area	m^2	σ_k, σ_ε	Turbulent Prandtl numbers	-
B	average volumetric concentration	-	L	Length of section	m
Q	Volumetric rate of flow	m^3/s	R_b	Radius of bubble	m
j	Volumetric flux	m/s	R_0	Diameter of orifice (air supplier)	m
σ	Surface tension	N/m	f, g	Fluid and gas phases	-