The Iraqi Journal For Mechanical And Material Engineering, Special Issue (E)

# Numerical Simulation of Three Dimensional Water Flow Through Spray Nozzle

**Dr. Tahseen Ali Hussein Al- Hatab**
Professor
Electrochemical Engineering Department
University of Babylon

**Mr. Sahir Rakim Raheam Al- Muhanna**
Assistant Lecturer
Mechanical Engineering Department
University of Babylon

**Mr. Ali Nu'man Ibrahim Al- Tufaily**
Assistant Lecturer
Mechanical Engineering Department
University of Babylon

A three dimensional numerical simulation of turbulent water flow through air washer system nozzles of cylindrical junction with diameters of 2 mm, 3 mm, 4 mm, 5 mm respectively and of centered diameter with 4 mm was achieved by using **FLUENT** software. **k-ε** model was used in this study. The entrance region of the model was selected to ensure the constant velocity in the inlet gives results near real solution. Modeling includes the construction of a 3D model with the aid of **AUTOCAD** and **GAMBIT**. For the purpose of checking the applicability of the modeling and discovering the best location and diameter of cylindrical junction, five different locations and diameters of the cylindrical junction were selected. The present work shows that the best location and diameter are observed for the case of 4 mm. The flow simulation was achieved by studying the velocity vector plot and contour plot of pressure in different slices locations through each of 5 geometries. The results were identical to the physical phenomenon which was observed in the laboratory of fluid mechanics, where the velocities and pressures are zero at exit cylinder center which means that there is no fluid at center of exit cylinder for 4mm case as shown in Fig(1). The results show also that the maximum skin friction occurs at the peripheral boundaries of the cylindrical junction and exit cylinder.


**Fig.(1) Air Washer of the present work shows the vortex in the center of cylinder with no fluid.**

**الخلاصة:-**

تم إنجاز نمذجة عددية ثلاثيّة الأبعاد لإنسياب الماء المضطرب خلال منفث ترذيذ منظومة غسيل الهواء ذات وصلة اسطوانية بأقطار 2 ملم, 3 ملم, 4 ملم على التعاقب و وصلة اسطوانية ذات قطر 4 ملم متمركزة مع أسطوانتي المنفث باستخدام برنامج ال (FLUENT). أستخدم في هذه الدراسة نموذج **k-ε** كنموذج للاضطراب. تم إطالة منطقة المدخل لكي يُضمن كون سرعة المدخل الثابتة تعطي حل قريب من الواقع. تضمَّنَت النمذجة بناء **نموذج ثلاثي الأبعاد** باستخدام برنامجي ال **(AUTOCAD)** وال**(GAMBIT)**. لغرض فحص قابلية النمذجة على التطبيق مع النموذج واستكشاف الموقع والقطر الأفضل للوصلة الأسطوانية تم اختيار خمس مواقع وأقطار لهذه الوصلة. بيّنت نتائج العمل الحالي بأن الموقع والقطر الأفضل يُلاحظان للحالة ذات 4 ملم. تم إنجاز نمذجة الإنسياب بدراسة مُخطَّطي مُتّجه السرعة والضغط الكنتوري عند مواقع مختلفة لشرائح مقطعيّة خلال الخمس حالات. كانت النتائج مُطابقة للحالة الفيزياويّة التي لوحظت في مختبر الموائع حيث تكون قيم كلٍ من السرع والضغوط صفراً عند مركز أسطوانة المخرج وهذا يعني عدم وجود للمائع في مركز أسطوانة المخرج للحالة ذات 4 ملم كما في الشكل (1). بينت النتائج بأن الاحتكاك السطحي الأعظم يحدث خلال حدود الوصلة الأسطوانية وعلى الجدران المحيطيّة لأسطوانة المخرج.

## 1. Introduction

Air washer spray nozzle is the most important part in many A/C systems, where the efficiency of these systems is evaluated according to this part efficiency. It plays the active role in smallest water droplets spray spread through the hot air to ensure the large rate of heat absorption from the last. When small drop of water meets the hot air surface it absorbs the required latent heat of evaporation. By increasing the rate of evaporation, the rate of heat removal from air will also increased. Finally the result will be air with low temperature and appropriate amount of moisture for conditioning purposes.

In spite of existing many types of air washers, they conjoin in having small cylindrical junction (see fig.(2)).

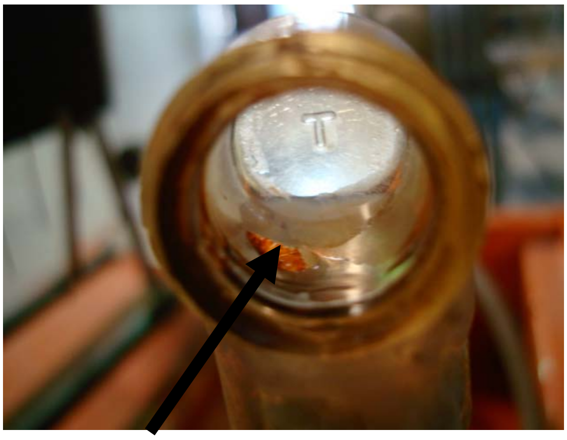


**Fig.(2) Opening of the cylindrical junction.**

The determination of the best location of the cylindrical junction is too difficult and it is of great importance to achieve the best function for the spray nozzle of the air washer system. Till now, there was no research found to achieves this task, hence the present paper provides a mathematical modeling for simulating the flow through nozzle of air washer system and studying different locations of cylindrical junction. The original cylindrical junction diameter is 2 mm, and the diameters of the cylindrical junction which were selected as the models for studying the flow through the spay nozzle are 3mm, 4mm, 5mm and 4mm at the center of the long cylinder which is connected with the cylindrical junction. There were no results for comparison, hence the results of the present work was checked by compare it with practical observation. the results of the present model were so identical with those noticed in the laboratory.

## 2. Mathematical Model

The governing equations for fully developed turbulent flow, steady, incompressible, with dissipation function and neglected body forces and buoyancy effects of water through spray nozzle are as reported in [1] , as follows:

**Continuity :-**

$$\frac{1}{y}\frac{\partial}{\partial y}(yu)+\frac{1}{y}\frac{\partial}{\partial z}(v)+\frac{\partial}{\partial x}(w)=0 \qquad \dots (1)$$

**Momentum Equations:-**

**Momentum Equation in Radial Coordinate ( y ):**

$$\rho\left(u\frac{\partial(u)}{\partial y}+\frac{1}{y}\frac{\partial(v)}{\partial z}+v\frac{1}{y}\frac{\partial(u)}{\partial z}-\frac{v^2}{y}+w\frac{1}{y}\frac{\partial(u)}{\partial x}\right)=-\frac{\partial\overline{P}}{\partial y}+\left[\frac{1}{y^2}\frac{\partial}{\partial y}\left(y^3\mu_{eff}\frac{\partial}{\partial y}\left(\frac{u}{y}\right)\right)+\frac{1}{y}\frac{\partial}{\partial z}\left(\frac{\mu_{eff}}{y}\frac{\partial u}{\partial z}\right)+\frac{\partial}{\partial x}\left(\mu_{eff}\frac{\partial u}{\partial x}\right)-\mu_{eff}\frac{2}{y^2}\frac{\partial v}{\partial z}\right]+S_y \qquad \dots (2)$$

Where

$$S_y=\frac{1}{y^2}\frac{\partial}{\partial y}\left(y^3\mu_t\frac{\partial}{\partial y}\left(\frac{u}{y}\right)\right)+\frac{1}{y}\frac{\partial}{\partial z}\left(\mu_t y\frac{\partial}{\partial y}\left(\frac{u}{y}\right)\right)+\frac{\partial}{\partial x}\left(\mu_t\frac{\partial w}{\partial y}\right)-\frac{2}{3}\rho\frac{\partial k}{\partial y} \qquad \dots (3)$$

**Momentum Equation in Angular Coordinate (z):**

$$\rho\left(u\frac{\partial(v)}{\partial y}+\frac{v}{y}\frac{\partial v}{\partial z}+\frac{uv}{y}+w\frac{\partial v}{\partial x}\right)=-\frac{1}{y}\frac{\partial\overline{P}}{\partial z}+\left[\frac{1}{z^2}\frac{\partial}{\partial z}\left(y^3\mu_{eff}\frac{\partial}{\partial y}\left(\frac{v}{y}\right)\right)\frac{1}{y}\frac{\partial}{\partial z}\left(\frac{\mu_{eff}}{y}\frac{\partial v}{\partial z}\right)+\frac{\partial}{\partial x}\left(\mu_{eff}\frac{\partial v}{\partial x}\right)+\mu_{eff}\frac{2}{y^2}\frac{\partial u}{\partial z}\right]+S_z \qquad \dots (4)$$

Where

$$S_z=\frac{1}{y}\frac{\partial}{\partial z}\left(\frac{\mu_t}{y}\frac{\partial v}{\partial z}\right)+\frac{2}{y}\frac{\partial}{\partial z}\left(\mu_t\frac{u}{y}\right)+\frac{\partial}{\partial y}\left(\frac{\mu_t}{y}\frac{\partial u}{\partial z}\right)+\frac{\partial}{\partial x}\left(y\frac{\mu_t}{y}\frac{\partial w}{\partial z}\right)-\frac{2}{3}\frac{\rho}{y}\frac{\partial k}{\partial z} \qquad \dots (5)$$

**Momentum Equation in Axial Coordinate (x):**

$$\rho\left(u\frac{\partial w}{\partial y}+\frac{v}{y}\frac{\partial w}{\partial z}+w\frac{\partial w}{\partial x}\right)=-\frac{\partial\overline{P}}{\partial x}+\left[\frac{1}{y}\frac{\partial}{\partial y}\left(y\mu_{eff}\frac{\partial w}{\partial y}\right)+\frac{1}{y}\frac{\partial}{\partial z}\left(\frac{\mu_{eff}}{y}\frac{\partial w}{\partial z}\right)+\frac{\partial}{\partial x}\left(\mu_{eff}\frac{\partial w}{\partial x}\right)\right]+S_x \qquad \dots (6)$$

$$S_x=\frac{1}{y}\frac{\partial}{\partial y}\left(\mu_t y\frac{\partial u}{\partial x}\right)+\frac{1}{y}\frac{\partial}{\partial z}\left(\mu_t\frac{\partial v}{\partial x}\right)+\frac{\partial}{\partial x}\left(\mu_t\frac{\partial w}{\partial x}\right)-\frac{2}{3}\rho\frac{\partial k}{\partial x} \qquad \dots (7)$$

The (k-ε) model in terms of cylindrical coordinates can be expressed as follows :

The kinetic energy of turbulence equation, k

$$\rho\left(u\frac{\partial k}{\partial y}+\frac{v}{y}\frac{\partial k}{\partial z}+w\frac{\partial k}{\partial x}\right)=\left[\frac{1}{y}\frac{\partial}{\partial y}\left(y\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial y}\right)+\frac{1}{y}\frac{\partial}{\partial z}\left(\frac{\mu_t}{\sigma_k}\frac{1}{y}\frac{\partial k}{\partial z}\right)+\frac{\partial}{\partial x}\left(\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x}\right)+G-\rho\varepsilon\right] \quad \ldots(8)$$

The dissipation rate of kinetic energy of turbulence equation, ε

$$\rho\left(u\frac{\partial \varepsilon}{\partial y}+\frac{v}{y}\frac{\partial \varepsilon}{\partial z}+w\frac{\partial \varepsilon}{\partial x}\right)=\left[\frac{1}{y}\frac{\partial}{\partial y}\left(y\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial y}\right)+\frac{1}{y}\frac{\partial}{\partial z}\left(\frac{\mu_t}{\sigma_\varepsilon}\frac{1}{y}\frac{\partial \varepsilon}{\partial z}\right)+\frac{\partial}{\partial x}\left(\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x}\right)\right]$$

$$+C_{\varepsilon 1}\frac{\varepsilon}{k}G-C_{\varepsilon 2}\rho\frac{\varepsilon^2}{k} \quad \ldots(9)$$

The Generation term, G in equations (8) and (9) can be written as follows:-

$$G=\mu_t\left\{2\left[\left(\frac{\partial u}{\partial y}\right)^2+\left(\frac{1}{y}\frac{\partial v}{\partial z}\right)^2+\left(\frac{\partial w}{\partial x}\right)^2\right]+\left(\frac{1}{y}\frac{\partial w}{\partial z}+\frac{\partial v}{\partial x}\right)^2+\left(\frac{\partial u}{\partial x}+\frac{\partial w}{\partial y}\right)^2\right.$$

$$\left.+\left(\frac{1}{y}\frac{\partial u}{\partial z}+\frac{\partial v}{\partial y}-\frac{v}{y}\right)^2\right\} \quad \ldots(10)$$

**3. <u>Numerical Scheme</u>**

The numerical simulation of any **3D** model is so complicated, especially when the geometry of the model is very complex. The complexity comes from many causes like building and meshing the model, converting the governing equations into appropriate algebraic form consistent with the program which prepared for solving it, …..etc [2,3]. The task will be simpler, when one treats with software, but this treating requires precise description of the problem. Many numerical packages and software may be used to solve the CFD problems such as ANSYS and FLUENT [4,5,6]. In this work the FLUENT witch used the FVM approach is used to solve the problem numerically. The present study was achieved by drawing manycomplicated **3D** figures of whole air washer nozzle with different locations of cylindrical junction by **AUTOCAD**, meshing all figures (each one alone) by **GAMBIT** software, applying the boundary conditions and then solving the problem for each figure by **FLUENT**. The most important steps of numerical solution may be summarized as follows:-

I. **Building The Model:-** The difficulty of the solution of the problem resides in that it is three dimensional and the geometry is complex. The complexity of the geometry was exceeded by dividing it to three parts, two of them are two cylinders connected by the third part which is the cylindrical junction. The cylinder before the cylindrical junction will be called inlet cylinder and after it will be called exit cylinder. The drawing of each part was accomplished by **AUTOCAD** and the final step of building the model is joining the three parts. The aim of dividing the geometry as addition to simplify it, is meshing it in three dimensions. The entrance length of the flow was lengthen to to ensure the constant velocity boundary condition in the inlet gives results near real solution. Figure (3) represent the geometry of the problem and the other geometry differs in cylindrical junction diameter, one of these has centered position, The distance between the inlet cylinder center and the center of cylindrical junction is (33mm) .

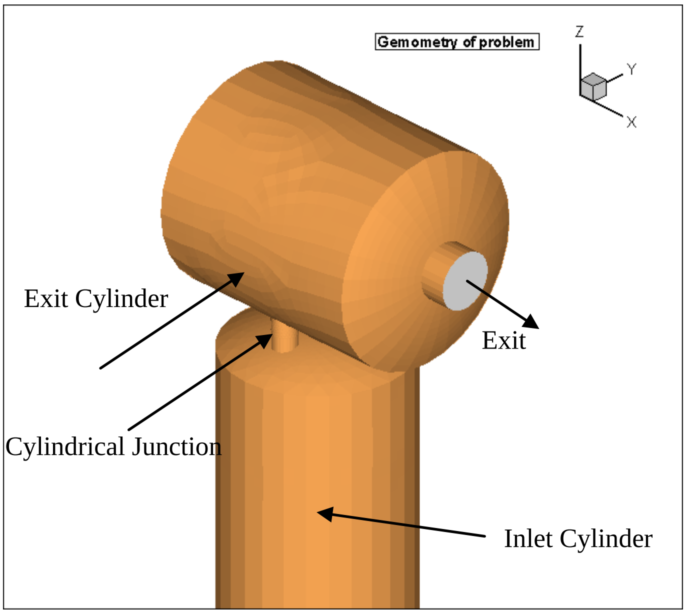
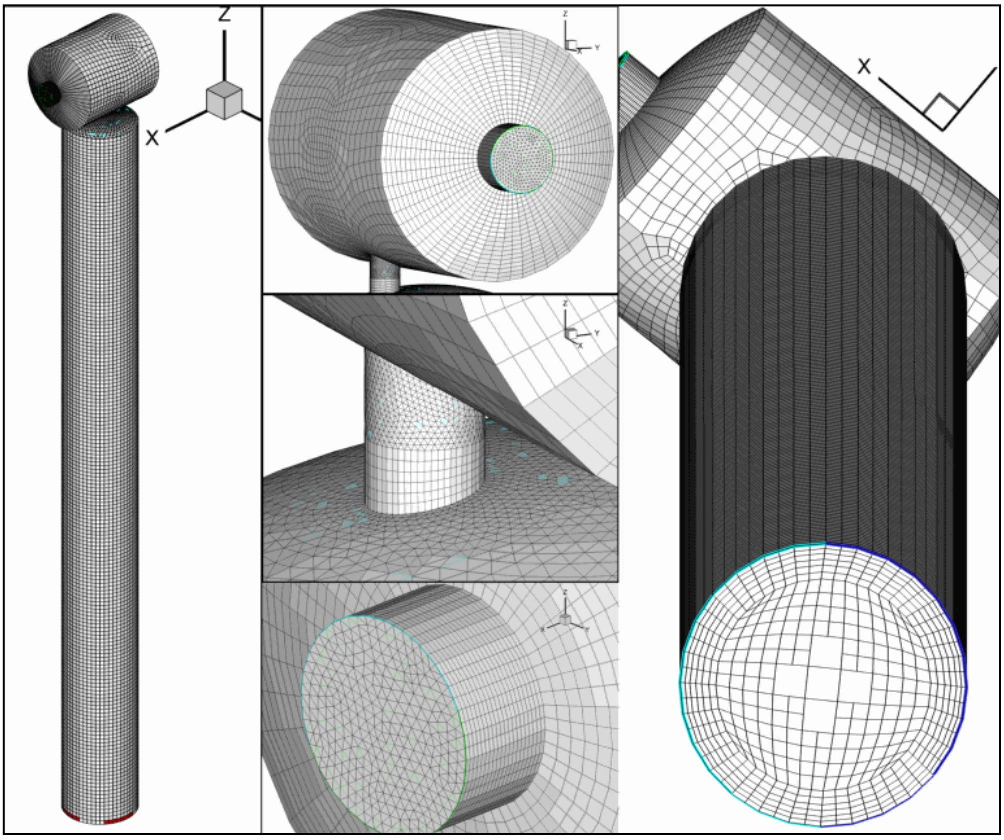



**Fig.(3) Geometrical Shape drawn in AUTOCAD.**

**II.** **Meshing The Model:-** It was done by using **GAMBIT** (Geometry And Mesh Building Intelligent Toolkit) [2] (2005). **GAMBIT** is a program prepared for previous treatment of drawing and meshing the geometry to be valid for treating it with **FLUENT**. Many types of elements were selected according to the type of boundaries to mesh the three parts of the geometry. The two cylindrical parts were meshed with hexahedral elements, but the cylindrical junction was meshed with tetrahedral elements. These types of elements were selected to coincide the boundaries, to make the boundaries effective. The meshed geometry can be seen in figure (4).



**III.** **Applying The Boundary conditions:-** Since the temperature was assumed to be constant in the flow of this problem, there was needing for boundary conditions related to velocities and pressures; and since the flow is turbulent and modeled with **k-ε** model , hence the velocities, **k** and **ε** values at inlet and exit should be specified as addition to considering the pressure at exit to be atmospheric. The value of flow rate was calculated at the fluid laboratory for spray nozzle of air washer system to be 4.98893E-05 $cm^3/s$. from it , Reynolds number values at inlet and exit were calculated to be 4358 and 14042.6 respectively. Turbulence values at inlet and exit were calculated by depending on calculating the turbulence intensity, I which calculated according to the following empirical relation [3] (2001):-

$$I=0.16\ Re^{-1/8} \qquad \text{……(11)}$$

## 4. Experiment Labrotary.

a simple experiment was done in fluid laboratory the tools of experiment was hydraulic bench, pipes and air washer nozzle as shown in Fig(1), this type of nozzle was used for air cooler in the engineering college. The volume flow rate of water was changed in order spraying water in air, the water is accumulated in the tank of hydraulic bench by making the nozzle facing the ground of tank and compute the time required for accumulate 2 liters in order computing the volume disgharge.

## 5. Result and Discussion

It is so troublesome to show all 3D results clearly, therefore the results of the present study will be displayed as 2D results by selecting certain slices on specified planes (in which the flow has severe changes in physical properties).

Figure (5) shows the variations of the pressure contours at **y=0** plane for spray nozzles of cylindrical junction with diameters of 2 mm, 3 mm, 4 mm, 5 mm respectively and of centered diameter. These will be called the five cases. From this figure it is clear that there are variations in pressure for flow thought the five cases, the first one is the difference in the pressure at the inlet cylinder, The variation appears from red to blue color (red is higher approximately 70 kpa gage pressure and the blue is lower approximately 14.60 kpa gage pressure), as the diameter of cylindrical junction increases the pressure in the inlet cylinder decreases that is mean higher pump head is needed for case (1) and smaller pump head is needed for case (4),the second variation of the pressure contour in the head of spray nozzle is the gradient of pressure from the center to the peripheral of exit cylinder, actually these gradients of pressure refers to the phenomenon of forced vortex, in case (3) & (4) the uniform gradient is observed with respect to other cases.

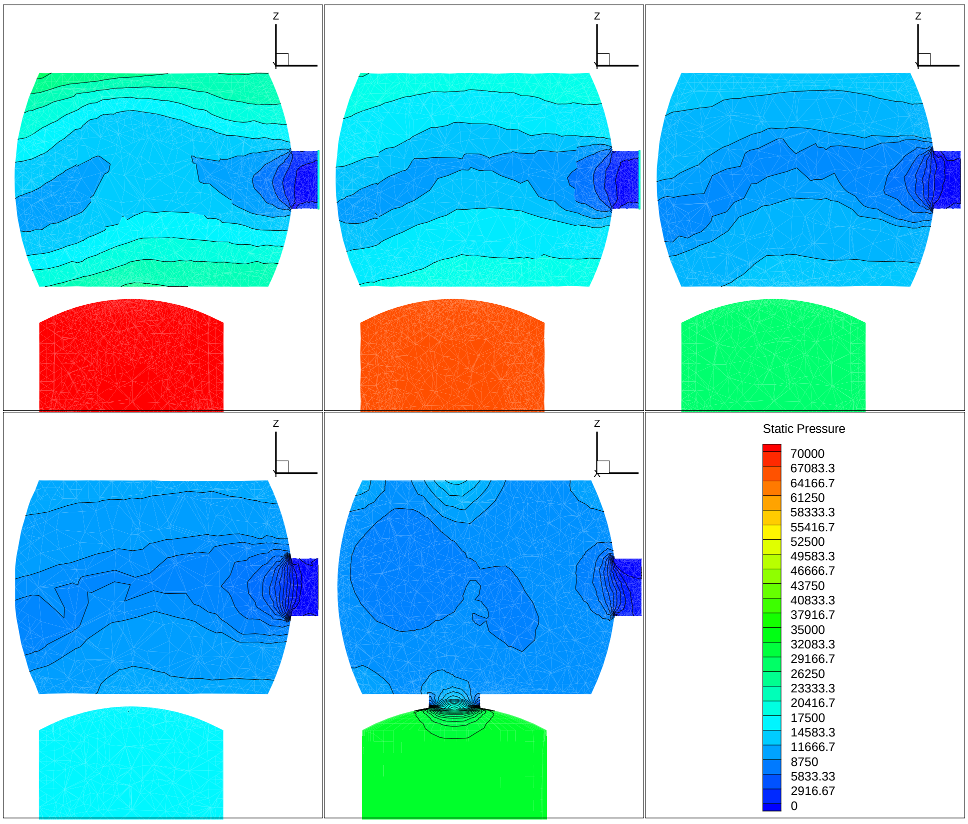
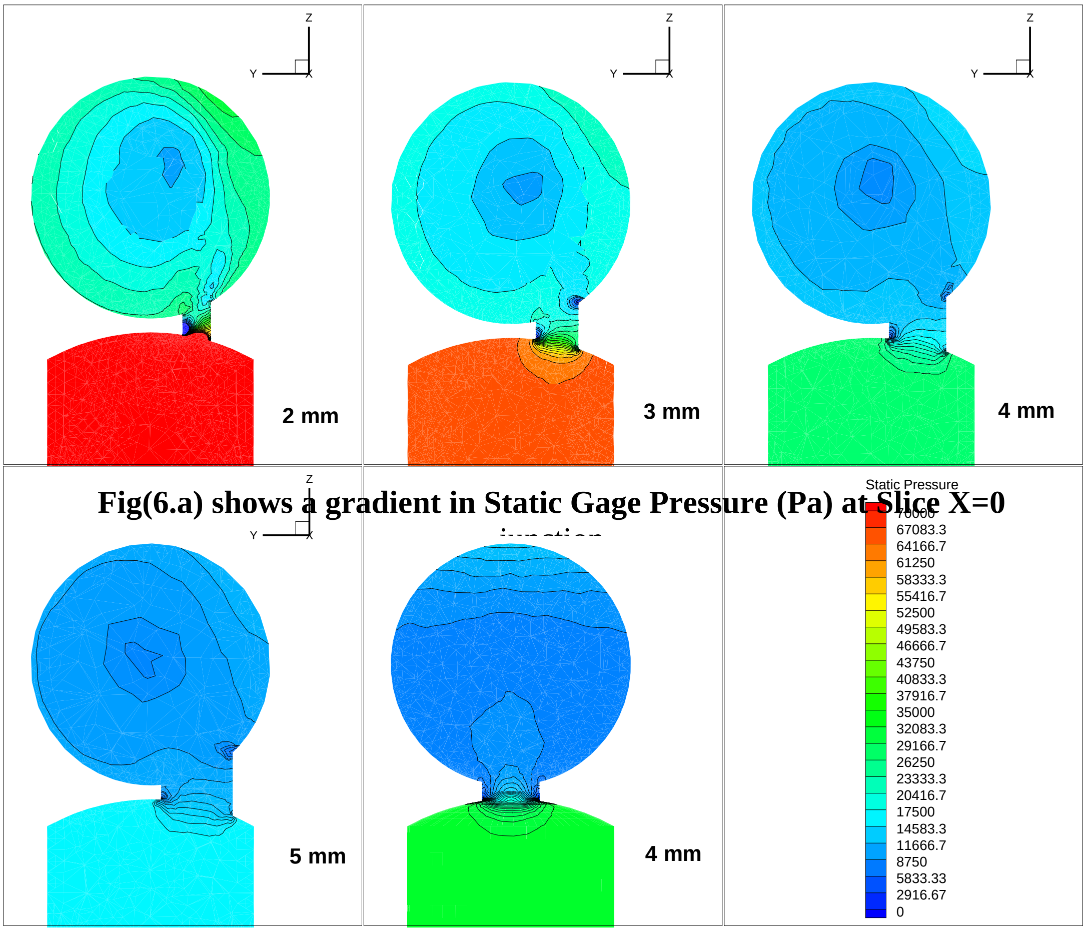



**Fig.(5) Variations of Static Gage Pressure (Pa) Contours at Slice Y=0 .**

Figure (6.a) shows the same facts which were analyzed from figure (5) but at x plane, and one can add that the worse case is of case(5)(the phenomenon isn't founded) because the pressure gradient isn't around the center of the exit cylinder.




Fig(6.a) shows a gradient in Static Gage Pressure (Pa) at Slice X=0 junction

Fig(6.b) Shows a Great Gradient In Static Gage Pressure (Pa) in The Cylindrical Junction

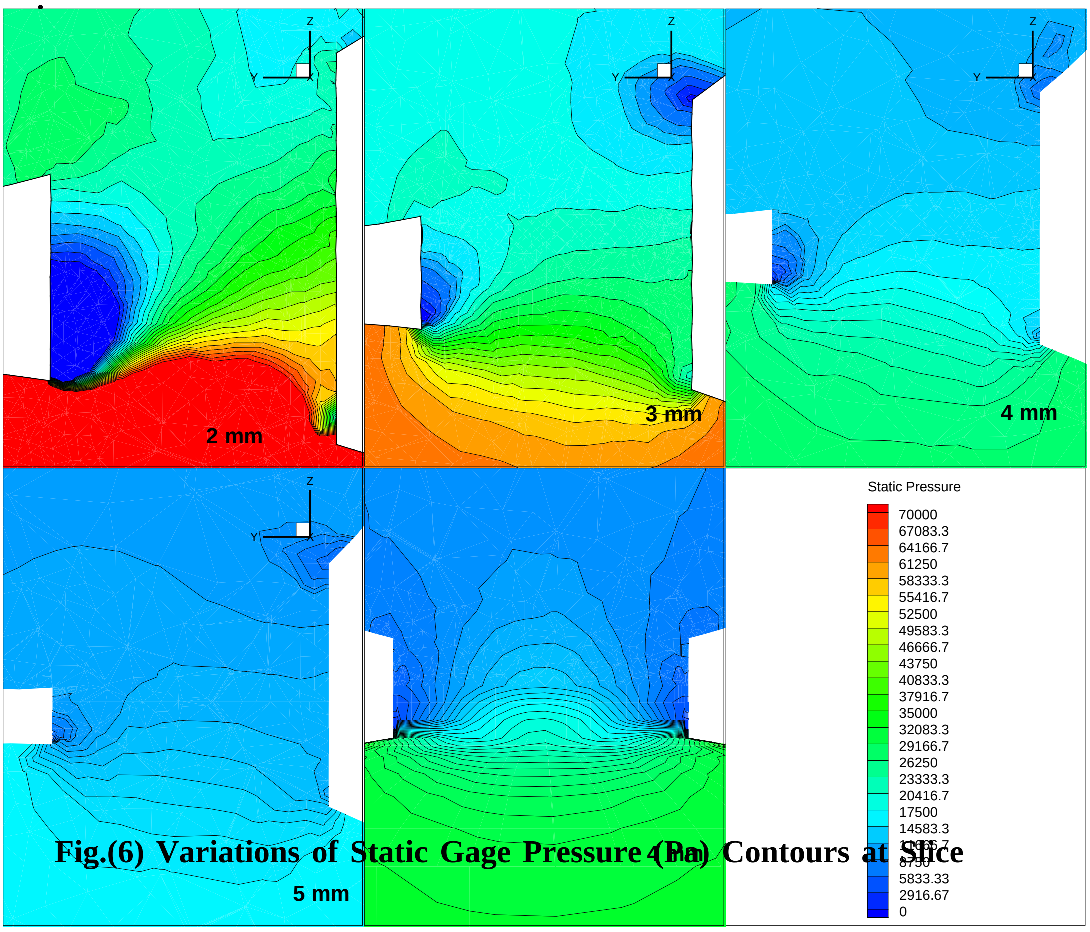



Fig.(6) Variations of Static Gage Pressure (Pa) Contours at Slice

Fig(6.b) shows a great gradient in pressure in the cylindrical junction, And those effects comes from the differences of the diameter of the junction, as the diameter decreases the velocity increases and the disturbance increases in regions downstream of higher deviation in direction of flow, this deviation causes pressure drop and reverse flow, those regions appears in higher degree of blue such as case(1)&case(2), that is mean higher losses on them.

Figure (7.a) shows the velocity vector of five cases at y plane. From this figure, it is noted that the minimum velocities (zero velocities) occur at the region near centers of exit cylinders of case (1),(2),(3),(4) , the vortex motion occurs around the center of exit cylinder. As the velocity vector to be fitted to the boundary or parallel to it, that is mean little losses and best case, also center of exit cylinder coincide with center of vortex or near coincidence , Since the fluid moves during the flow, the solely interpretation of zero velocities is that there is no fluid in the central region and this demonstrate that the present modeling closes from the actual situation shown in fig(1)because that the circular motion of fluid particle needs the centrifugal force this force gives the shape of vortex . The case of 5 mm has the larger velocities at exit cylinder center, hence it is the farthest case of spraying water. All cases have steep velocity variations at the cylindrical junction because of sudden contraction of area in this junction. The case of 3 mm has also large values of velocities at exit cylinder center.

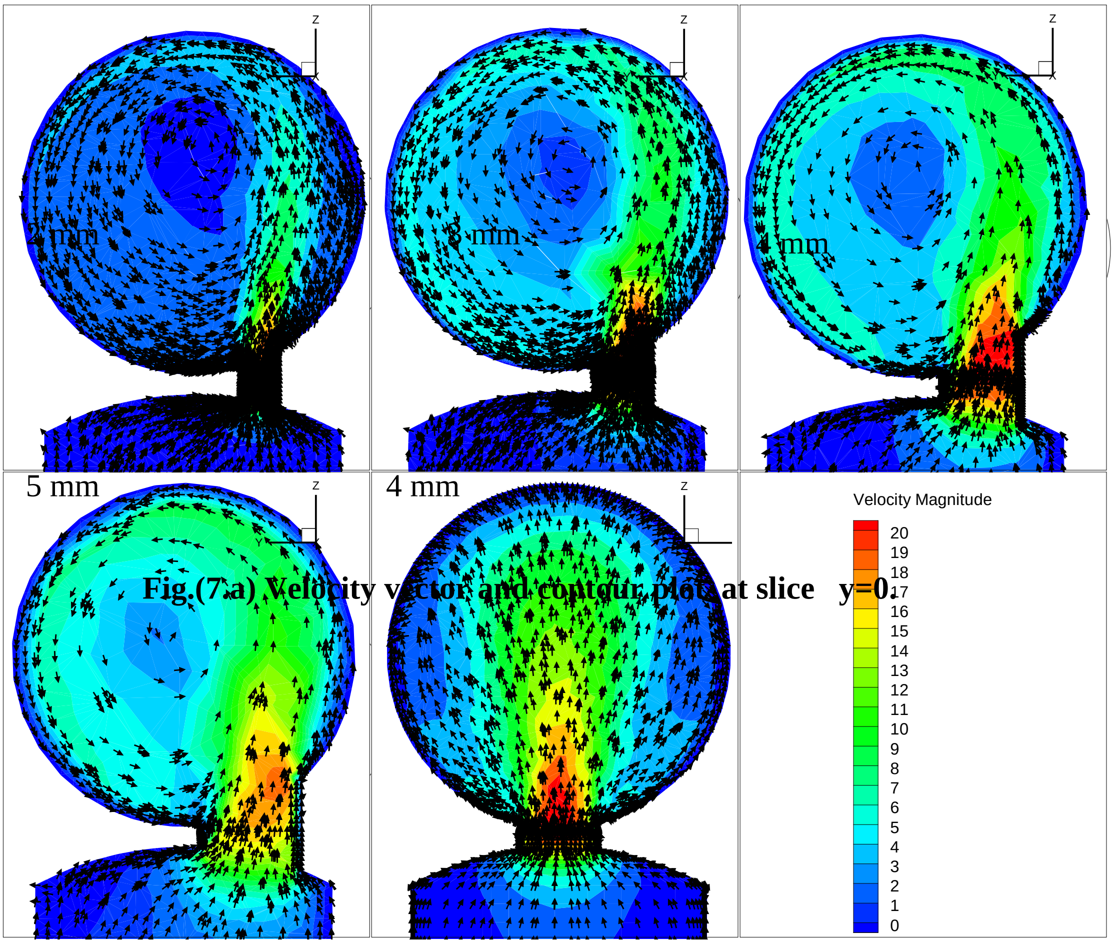
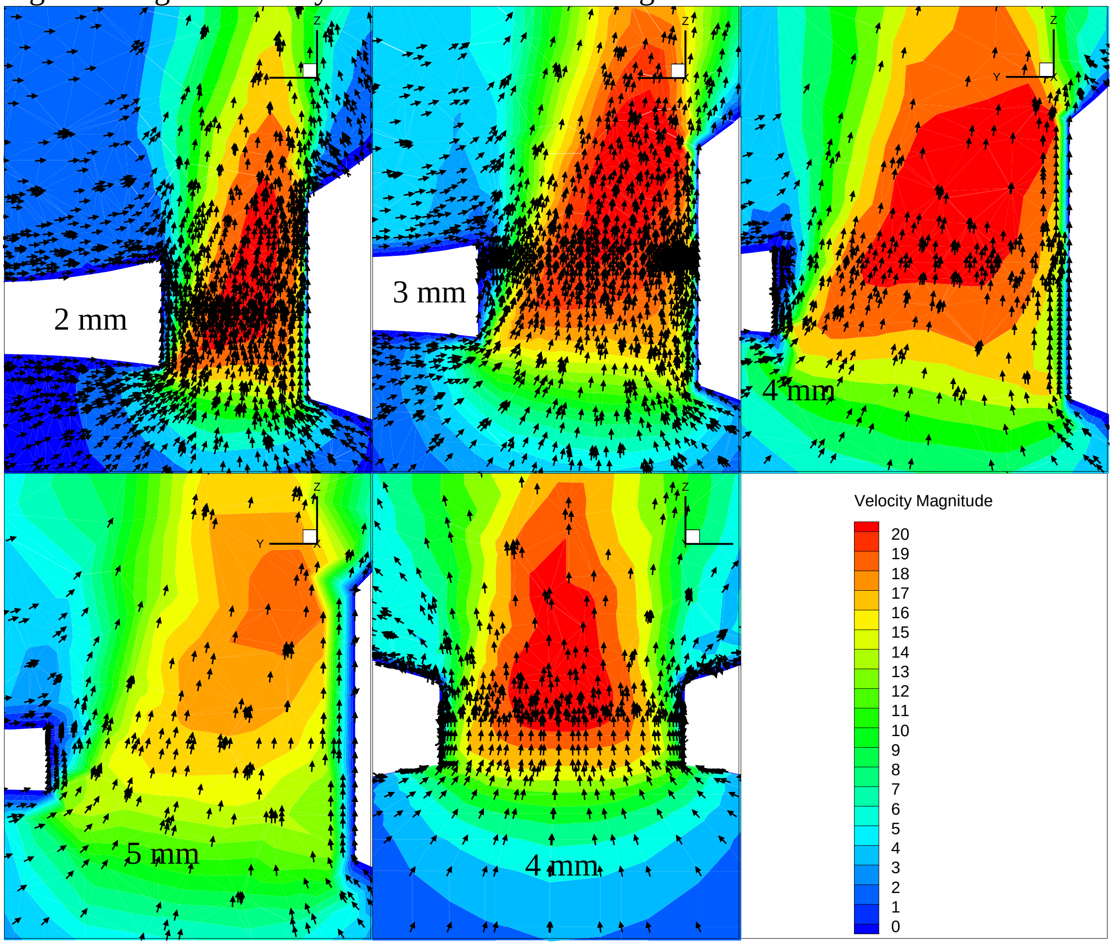



**Fig.(7.a) Velocity vector and contour plot at slice y=0.**

Fig(7.b) shows the velocity vector at cylindrical junction (note:- the vector length style refers to the direction of flow only because it is impossible to plot it because of higher gradient in velocity, and contour plot refers to velocity magnitude) the magnitude of velocity near the boundary is small and closes to zero ,also the flow is reversed in the higher change of velocity direction as shown in figure.




**Fig(7.b) Velocity Vector at Cylindrical Junction.**

Figure (8) shows the velocity vector of five cases at x plane. Three slices was taken to show the profile of vortex, if we sketch an imaginary curve starting from the first slice to third slice we will construct the shape of vortex, if the profile of vortex coincide with line pass through the center of exit cylinder and parallel to X-axis and with direction of jet as shown. Therefore the case(5) doesn't give flow the ability to spraying water at exit because there is no vortex motion and no centrifugal acceleration required for spray process.

Vortex motion phenomenon occurs in the case(1),(2),(3),(4) but case(1),(2) have more losses and higher pressure difference is required, case (3) has good pressure gradient and little losses with respect to case (4), case (5) wasn't achieved the phenomenon hence the better case for spray nozzle is case(3) that has 4 mm diameter of cylindrical junction.

There is no fluid at the centre region of the exit cylinder of case(1),(2),(3),(4) because the velocity and pressure values are zero at that region and centrifugal acceleration applied to the fluid particle required centrifugal force, this force throws fluid particle in radial direction of exit cylinder.

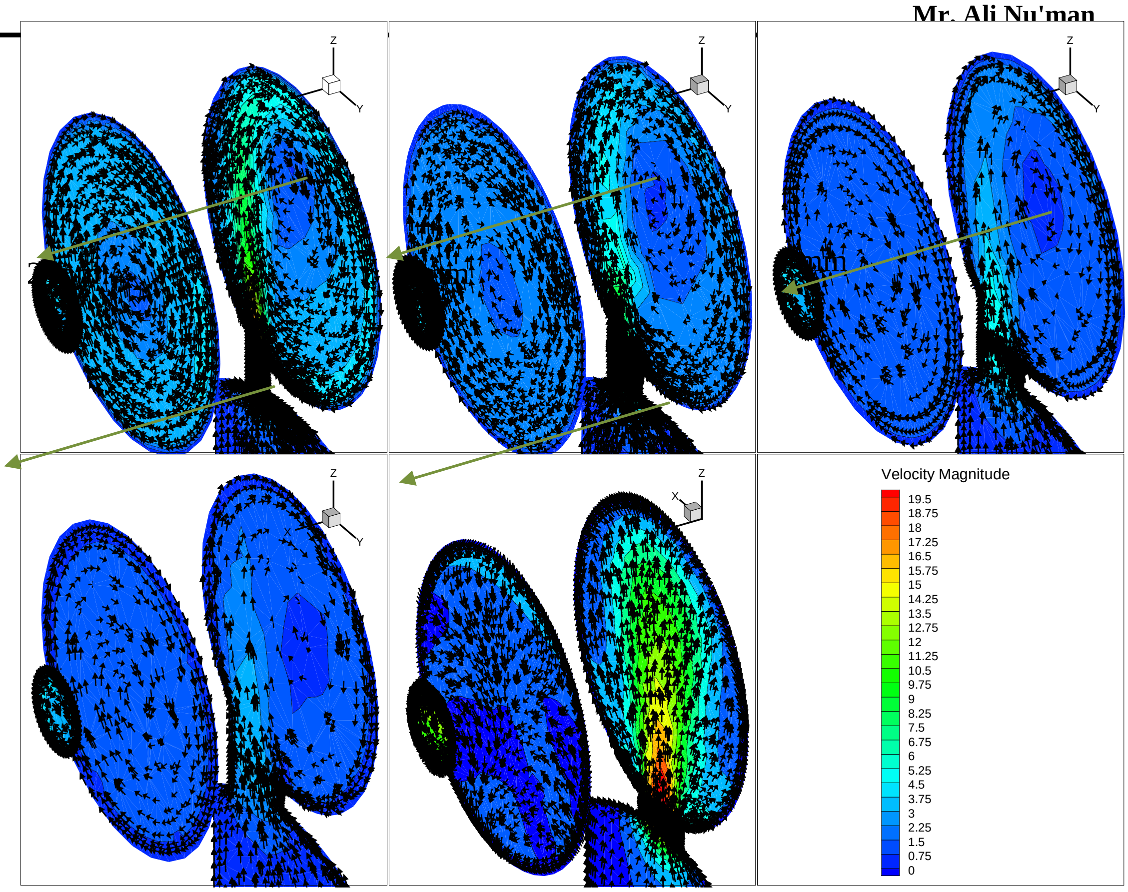



**Fig.(8) Velocity Vector at different slices.**

Fig(9.a)shows The skin friction values of the five cases in x plane, contour plots color were cycled twice and the changing values are exponentially in order to cover the great gradient in skin friction , contour plots of skin friction at x plane were depicted on Fig(9.a)and Fig(9.b) respectively . It has very high values near the boundaries of the exit cylinder and at the junction because of friction between the water particles of high velocities and the peripheral boundaries and the random turbulent motion of flow eddies. At the boundaries the effects of viscosity appear clearly and this is checked from the high skin friction values in spite of high velocities at these regions under the effect of swirling. The skin friction for all cases have the maximum values near walls and it decrease away from walls till reaching the maximum thickness of boundary layer where no viscosity effects.

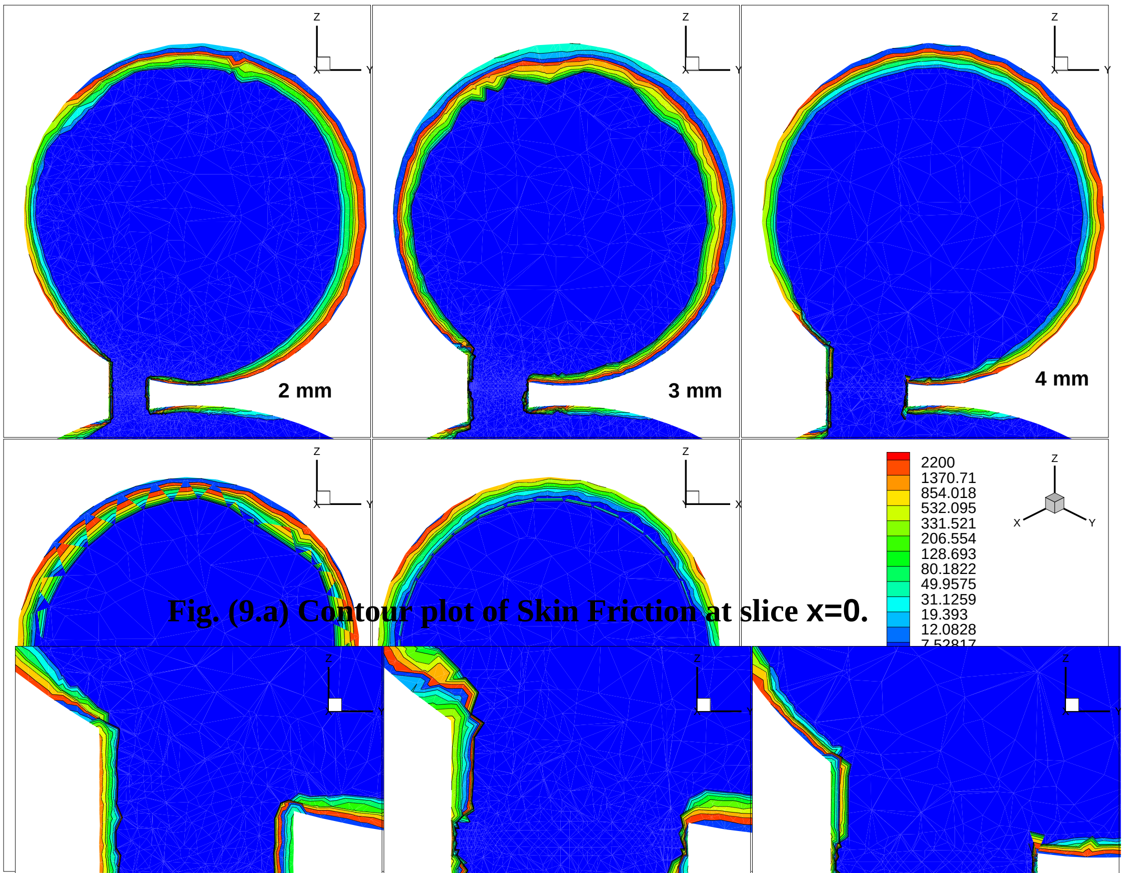



**Fig. (9.a) Contour plot of Skin Friction at slice x=0.**

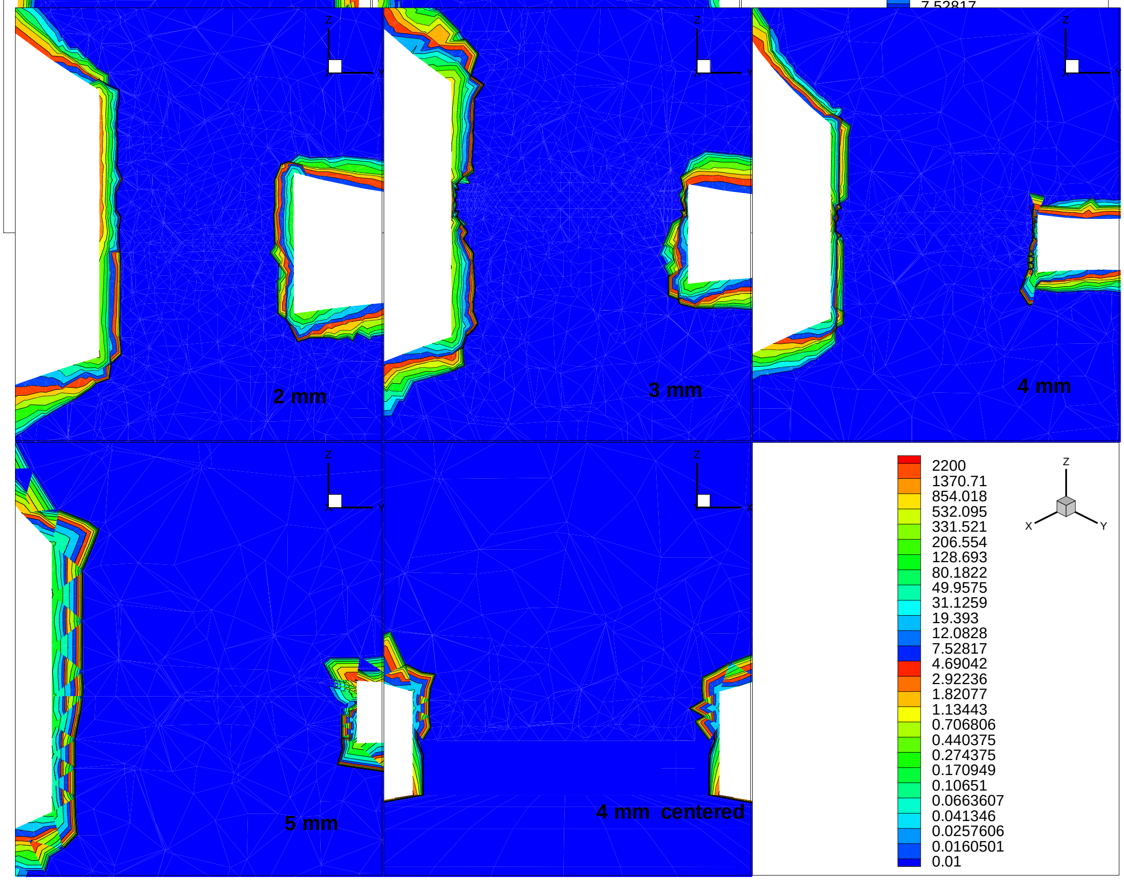



**Fig. (9.b) Contour plot of Skin Friction at cylindrical junction.**

**5.**

The ability of air washer system nozzle to spray water more efficiently can be checked for any configuration by checking the flow behavior through it. The swirling of the water around the peripheral boundaries with the central region of the exit cylinder with no fluid ensures effective spraying effect. Therefore, the case of 4 mm (case(3)) has the best ability of spraying. There is no benefit of make the cylindrical junction diameter more than 4 mm, and it is useless to make the cylindrical junction at the center of the inlet cylinder. This case is identical with the spray nozzle shown in Fig(1) in fluid laboratory.

The mathematical model for the present work succeed in simulating the three dimensional flow of water with very good results where it shows that the velocities at the central region of the exit cylinder has velocities with zero values and this means that there is no fluid at that region (these results coincide the physical situation viewed in the laboratory during operating the spray nozzle).

The skin friction values were very high especially at the boundaries of the cylindrical junction and exit cylinder because of the high velocities of the flow and high random motion of turbulent eddies, these high values form very exorbitant losses of the main flow and the pump which operates the system should retrieve this lack in energy.

## 6. References:-

**1**. Ayad Mahmoud Salman, (2003), “ Turbulent Forced Convection Heat Transfer in The Developing Flow Through Concentric Annuli, ” M.Sc. Thesis ,Mech. Eng. Dept., University of Technology.

**2.** Fluent Inc. (2003) ,“Introduction to Gambit 2.2” Training Notes.

**3**. Fluent Inc. (2001) ,“ FLUENT 6 .0 User’s **Guide**”**,**volume **1**

**4**. Piero M. Armenante, Changgen Luo, Chun-Chiao Chou, Ivan Fort, Jaroslav Medek, Velocity profiles in a closed, unbaffled vessel: comparison between experimental LDV data and numerical CFD predictions. Chemical Engineering Science, Volume 52, Issue 20, October 1997, Pages 3483-3492

**5**. S. Murphy, R. Delfos, M.J.B.M. Pourquié, Olujiz, P.J. Jansens, F.T.M. Nieuwstadt Prediction of strongly swirling flow within an axial hydrocyclone using two commercial CFD codes. Chemical Engineering Science, Volume 62, Issue 6, March 2007, Pages 1619-1635

**6.** Matevz Dular, Rudolf Bachert, Bernd Stoffel, Brane Sirok. Influence of the velocity distribution at the inlet boundary on the CFD prediction of local velocity and pressure fields around a hydrofoil. Experimental Thermal and Fluid Science, Volume 32, Issue 3, January 2008, Pages 882-891

## Conclusions

**Nomenclatures**

| Symbol | Definition | Unit |
|---|---|---|
| $C_{\varepsilon 2}$ $C_{\varepsilon 1}$ | Turbulence Constants | — |
| G | Generation Term | $Kg/m.sec^3$ |
| K | Kinetic Energy of Turbulence per Unit Mass | $m^2/sec$ |
| P | Pressure | $N/m^2$ |
| S | Source Term | $N/m^3$ |
| u | Velocity Rate at Axial Direction | m/sec |
| v | Velocity Rate at Radial Direction | m/sec |
| w | Velocity Rate at Angularl Direction | m/sec |
| x | Axial Coordinate | m |
| y | Radial Coordinate | m |
| z | Angular Coordinate | m |

## Greek Symbols

| Symbol | Definition | Unit |
|---|---|---|
| ε | Dissipation Rate of Kinetic Energy of Turbulence per Unit Mass | $m^2/sec^3$ |
| μ | Viscosity | kg/m.sec |
| ρ | Fluid Density | kg/m3 |
| $\sigma_k,\sigma_\varepsilon$ | Turbulent Prandtl Number | |

## Subscripts

| Symbol | Definition | Unit |
|---|---|---|
| eff | Refers to Effective Viscosity | — |
| t | Refers to Turbulent Viscosity | — |
| x | Source Term in Axial Direction | — |
| z | Source Term in Angular Direction | — |

## Superscripts

| Symbol | Definition | Unit |
|---|---|---|
| – | Mean | — |