ANALYZE THE COMPRESSIBLE FLOW CHARACTERISTICS IN A CHANNEL WITH DOUBLE ELBOWS BY FINITE ELEMENT METHODS +

Ahmed Hassan Khuder *

Abstract :

Two–dimensional numerical simulation is carried out to study the fundamental characteristics of compressible flows through double elbow channel, the air used as a working substance. One channel include horizontal flow enters first elbow and the other include vertical entrance to the second elbow. In this paper, the flow fields in two elbows with their dead zone have been investigated. A non-linear k- Є model is employed to solve the flow characteristics by using the finite element method with a curvilinear coordinate system near the dead zone The k-Є model is a semi-empirical model based on model transport equations for turbulent kinetic energy k and its dissipation rate ε. i.e. the rate at which turbulent kinetic energy is converted into internal energy by viscous action. The derivation of turbulent kinetic energy and its rate of dissipation derived from Navier stocke equation. The model geometry and mesh were created using the COMSOL 3.5 software. This results of the simulations that have been made to date; baseline results employing the k-ε turbulence model are presented, the predicted value for the flow characteristics in double elbow is in reasonably good agreement with published correlations. For accurate solution the computation is considered converged to the steady solution, when the value of the maximal normalized equation residual is less than (10^{-6})· The model applied for two different flow velocities (20 m/s) and (30 m/s) and the obtained results presented as curves, surface and contours for velocities turbulent kinetic energy , rate of dissipation of turbulent kinetic energy and vortices .the builded model can be utilized for academic purpose since, it widely used were most flow systems contain elbows.

Key words : compressible flow, sudden change flow, flow in elbow

محاكاة عددية لتحليل خصائص الجريان الانضغاطي في قناة مزدوجة مع المرفقين بطريقة العناصر المحدودة

أحمد حسن خضر

المستخلص :

محاكاة عددية ثنائية الأبعاد تم توضيفها لدراسة الخصائص الأساسية للجريان في الموائع الانضغاطية في قناة ذات كوعين متعامدين ، تم استخدام الهواء كمادة الجريان الاساسية. القناة الاولى تشمل دخول التدفق الأفقي للجريان الى الأول للكوع والقناة الثانية تشمل مدخل عمودي للجريان الى الكوع الثاني. في هذا البحث ، تم دراسة وتحليل مجالات الجريان في القناة وفي الكوعين المتعامدين اضافة الى مناطق الضعف في الجريان. نموذج $k-\epsilon$

+ Received on 11/3/2014 , Accepted on 29/12/2014
* Assistant Lecturer / University of Technology

غير الخطي تم توضيفه بمحاكت عددية لحل خصائص الجريا ن باستخدام طريقة العناصر المحددة بنظام احداثيات خطي الا مناطق اطراف الكوع حيث تم استخدام احداثيات لا خطية . نموذج $k-\epsilon$ هو نموذج يعتمد على معادلات النقل للطاقة الحركية ا لمضطربة ,وعلى معدل فقدان هذه الطاقة. أي المعدل الذي يتم فيه تحويل الطاقة الحركية إلى طاقة داخلية مضطربة تحت تاثير لزوجة مدة الجريان .تم اشتقاق معادلات الطاقة الحركية المضطربة ومعدل تبديد الطاقة المستمدة من معادلة **Navier stocke**. تم بناءالنموذج الهندسي باستخدام برنامج . **COMSOL 3.5** هذه نتائج المحاكاة التي تم إجراؤها و عرض النتائج الأساسية التي تستخدم نموذج الاضطراب $k-\epsilon$ وخصائص الجريان في الكوع المزدوج كانت متطابقة مع البحوث العالمية المنشورة في هذا المجال. لغرض الحصول على دقة نتائج عالية تم تحديد قيمة الخطأ التراكمي المسموح به في البرنامج ب (10^{-6}) . النموذج الرياضي تم استخدامه بسرعتين مختلفتين الأولى (20 م/ثا) والثانية (30 م/ثا) والنتائج المستحصلة عرضت على شكل منحنيات وسطوح تتغير خلالها قيم السرع واللزوجة والطاقة الحركية المضطربة ومعدل فقدانها والدوامات. النموذج الذي تم دراسته وتحليله ممكن استثماره لاغراض اكاديمية كونه واسع الانتشار في العديد من المنظومات حيث ان معظم منظومات جريان الموائع انظغاطية او لاانضغاطية تتضمن وجود أكواع.

Nomenclature :

Symbols	Definition	Units
rho	Density	[kg/m^3]
eta	Viscosity	[Pa*s]
u	Inlet peak velocity	[m/s]
h_step	Step height	[cm]
L1	First elbow length	[m]
L2	Second elbow length	[m]
D	Channel diameter	[m]
N	Number of meshes element	-
Err	Accepted accumulated error	-

1. Introduction :

1.1 Backgrounds :

An elbow is a pipe fitting installed between two lengths of pipe or tubing to allow a change of direction, usually a 90° or 45° angle Fig. (1a), (1b). The ends may be machined for butt welding, threaded (usually female), or socketed, etc. When the two ends differ in size, the fitting is called a **reducing elbow** or **reducer elbow**. Bends are essential components of fluid flow systems, facilitating compact, lightweight designs. But they can also be the cause of complex secondary flows, losses and variable heat transfer rates in the fluid flow lines [1, 2]. Investigations of the flow through bends are of great significance in understanding and improving their performance and minimizing the losses. It is already well known that the flow of incompressible viscous fluids through pipe bends are characterized by flow separation, secondary flow and unsteadiness which are dependent on Reynolds number as

well as the radius of curvature of the bend. These in turn have significant influence on the total pressure loss and local heat transfer rate.
The compressible flow can choke in the bend with the possibility for strong oscillation due to secondary flow or separation. This can cause changes in the mass flow rate with time impacting the performance of the whole system. Accurate estimation of mass flow rate and losses are critical for most compressible flow systems. These factors call for a thorough understanding of the flow characteristics of compressible flow in pipe bends [3].

Figure (1.a): Long radius or sweep 90° elbow

Figure (1.b): Short radius or regular 45° elbow

Pipe fittings like valves, bends, elbows, tees, reducers, expander etc. are the integral part of any piping system. Flows through piping components are more complex than the straight pipes. The problem of determining the flow characteristics in elbows are important in design and analysis of the fluid machinery and many systems. Forcing a fluid through elbow consumes energy provided by the drop in pressure across the elbow.

1.2 Literature Review :

One of the main problems facing industries and designers on piping systems are the flow characteristics in elbow and curved paths **.**There are many studies conducted on the flow characteristics in elbow and piping systems. Some of these studies are theoretical and some are experimental.
The experimental solution requires measuring instrument with high accuracy and quick dynamic response during short time of operation. Theoretical treatment requires accurate initial and boundary conditions. The more related studies are: Researchers (Edwards *et al*. 1985; Das *et al.* 1991; Banerjee *et al.* 1994; Bandyopadhyay and Das 2007) [4, 5, 6, and 7] reported experimental studies of non-Newtonian liquid flow through various piping components and empirical correlation were suggested for individual piping components. However, data or equations for pressure drops through elbows are meager. Since most non-Newtonian liquids are highly viscous in nature and the laminar flow is of greatest practical interest (Das *et al*. 1989)[8] Two-phase gas-liquid flow through elbows is much more complex in nature. When flow enters the curved portion, the heavier density phase is subjected to a large centrifugal force, which causes the liquid to move away from the center of curvature. Studies on the effect of curve geometry on two-phase gas-liquid flow are sporadic in the literature Das et *al*. (1991); Banerjee and Das (1998); Bandyopadhyay *et al*. (2000)[9,10,11] reported the experimental investigation for gas-non-Newtonian liquid flow through bends, valves and elbows. They developed empirical correlation for predicting the

frictional pressure drop across the piping components. In order to achieve optimum performance, an accurate design technique is necessary for the prediction of the pressure drop for non-Newtonian and gas-non-Newtonian liquid through elbows.
Computational fluid dynamics (CFD) is also science can be employed for predicting fluid flow, heat and mass transfer, chemical reaction and related phenomena by solving numerically by the set of governing mathematical equations along with the conservation of mass, momentum, energy. The results of CFD analysis are relevant in conceptual studies of new designs, detailed product development, troubleshooting in existing unit and redesign. The CFD analysis complements testing and experimentation which reduces the total effort required in experimental design and data acquisition.
Computational Fluid Dynamics (CFD) can serve to evaluate the frictional pressure losses in piping systems, secondary flow effects can be visualized to aid in better understanding of the flow phenomena and can be applied to piping design for the improvement of the flow characteristics

2. Theoretical Algorithm :

The Navier-Stokes equations, which express the conservation of mass and momentum, and the transport equations for k and ε used in the turbulence model, form a coupled set of nonlinear partial-differential equations (PDEs). The COMSOL program uses a finite-element discretization to convert the PDEs to a set of nonlinear algebraic equations. The solutions obtained here employ the segregated solution algorithm, in which the equations are solved sequentially, as opposed to being assembled into a single matrix equation and solved simultaneously. Since the equations are nonlinear and coupled, the segregated method requires that an iterative process be used:
Starting from an initial “guess” for all the variables, the solution is updated, or allowed to “relax”, toward the final steady-state solution as the iterations proceed.
The inlet is used to specify the inlet velocity and outlet is used to specify pressure outlet. These geometries of the elbows are imported into COMSOL program in a Cartesian co-ordinate system. Turbulent non-Newtonian Power Law model have been used for simulation. The model solves for Navier-stokes equation at prescribes velocities. The governing equations are nonlinear and several iterations of loop must be performed before a convergent solution is obtained. The first-order upwind scheme is used in the discretization of set of governing equations, finite element schemes is used for calculating cell-face pressures for using the Segregated solver in COMSOL. Pressure-velocity coupling refers to the numerical algorithm which uses a combination of continuity and momentum equations to derive an equation for pressure (or pressure correction) when using the segregated solver.

The system of equations can be summarized by following basic equations

Equation for K

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \frac{\mu_t}{\rho} S^2 - \epsilon + \frac{\partial}{\partial x_j}\left[\frac{1}{\rho}\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]$$

Equation for ∈:

$$\frac{\partial c}{\partial t} + U_j \frac{\partial c}{\partial x_j} = \frac{c}{k}\left(C_{1c}\frac{\mu_t}{\rho} S^2 - C_{2c}\epsilon\right) + \frac{\partial}{\partial x_j}\left[\frac{1}{\rho}\left(\mu + \frac{\mu_t}{\sigma_c}\right)\frac{\partial c}{\partial x_j}\right]$$

Turbulent viscosity is calculated from:

$$\mu_t = \rho \, C_\mu \frac{k^2}{c}$$

There are 5 free constants:

$$\partial k \, , \partial_c \, , C_{1c} \, , C_{2c} \, , C_\mu$$

2.1 Overview of the equation for k :

- The equation for k contains additional turbulent fluctuation terms, that are unknown. Again using the Boussinesq assumption, these fluctuation terms can be linked to the mean flow..
- The following (simplified) model equation for k is commonly used

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_i}(\rho k \bar{u}_i) = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\partial_k}\right)\frac{\partial k}{\partial x_i}\right] + 2\mu_t \bar{S}_{ij} . \bar{S}_{ij} - \rho\varepsilon$$

Rate of Increase	Convective transport	Diffusive transport	Rate of Production	Rate of destruction

- The Prandtl number σ_k connects the diffusivity of k to the eddy viscosity. Typically a value of 1.0 is used.

2.2 Overview of the equation ϵ :

- A model equation for ϵ is derived by multiplying the k equation by (ϵ/k) and introducing model constants.
- The following (simplified) model equation for ϵ is commonly used

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial}{\partial x_i}(\rho\varepsilon\bar{u}_i) = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial\epsilon}{\partial x_i}\right] + C_{1\epsilon}\frac{\epsilon}{K} 2\mu_t \bar{S}_{ij} . \bar{S}_{ij} - C_{2\epsilon}\rho\frac{\epsilon^2}{K}$$

Rate of Increase	Convective transport	Diffusive transport	rate of production	Rate of destruction

- The Prandtl number σ_ϵ connects the diffusivity of ϵ to the eddy viscosity. Typically a value of 1.30 is used
- Typically values for the model constants $C_{1\epsilon}$ and $C_{2\epsilon}$ of 1.44 and 1.92 are used

2.3 Calculating the Reynolds stresses from k and ϵ :

– The turbulent viscosity is calculated from:

$$\mu_t = C_\mu \frac{k^2}{\epsilon} C_\mu = 0.09$$

– The Reynolds stresses are then calculated as follows:

$$-\rho\overline{u_i' u_j'} = \mu_t\left(\frac{\partial\overline{u}_i}{\partial x_j} + \frac{\partial\overline{u}_j}{\partial x_i}\right) - \frac{2}{3}\rho k\delta_{ij} = 2\mu_t\overline{S}_{ij} - \frac{2}{3}\rho k\delta_{ij}$$

– The $(2/3)\rho k\delta_{ij}$ term ensures that the normal stresses sum to k

Numerical treatment of the model :

Table no.1 and Figs. (2&3) represent steps of the main considerations that taken into account during modeling of compressible flow through the channel with double elbow.

Table (1): data used to solve the model

rho	1.23[kg/m^3]	Density
eta	1.79e-5[Pa*s]	Viscosity
u	20 &30[m/s]	Inlet peak velocity
h_step	3.81[cm]	Step height
L1	1 [m]	First elbow length
L2	1 [m]	Second elbow length
D	0.2 [m]	Channel diameter
N	5024	Number of meshes element
Err	10^{-6}	Accepted accumulated error

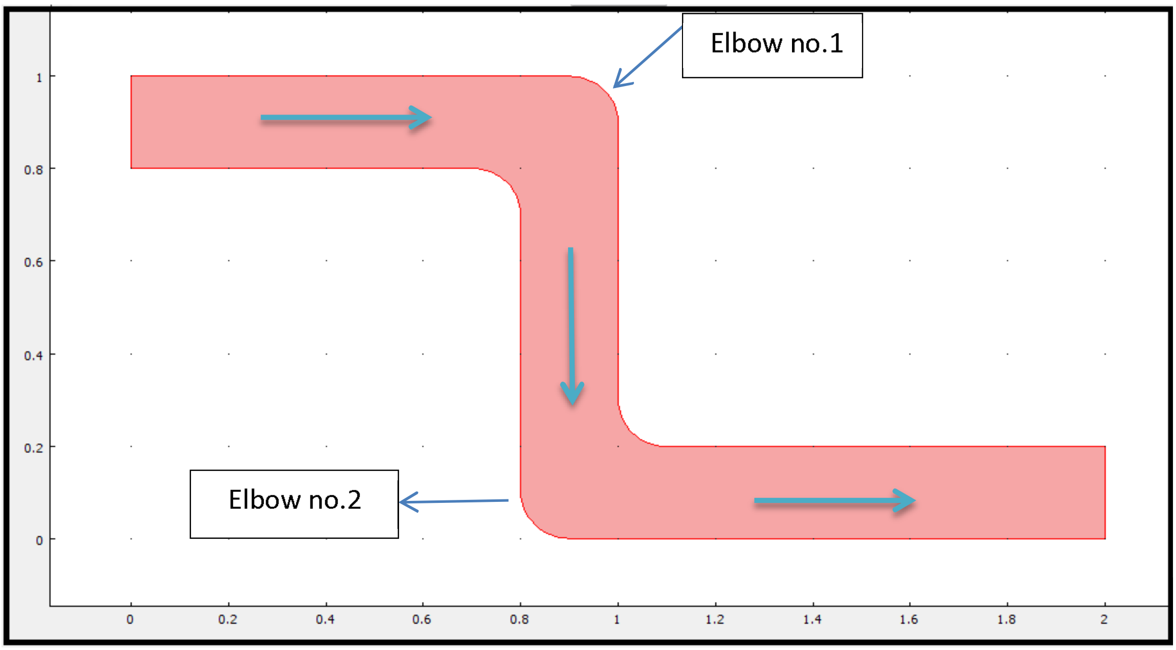

Figure (2): the geometrical shape of the channel with double elbow

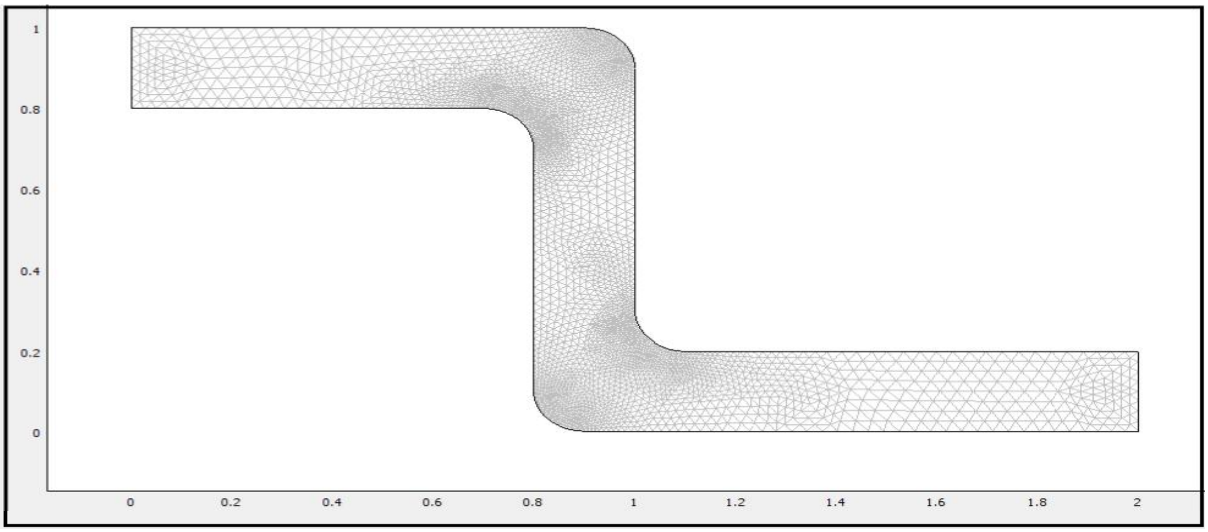

Figure (3): the physical domain of channel with mesh divisions

Results and discussion :

For the application of the mentioned theory of compressible flow through open channel with double elbow, a numerical solution for the geometry is presented.

The solution include the velocity profile through the channel with double elbow under initial velocity u = 20 m/s and 30 m/s, the velocity contours through the channel with double elbow under inlet velocity u = 20 m/s and 30 m/s, turbulent kinetic energy through the channel with double elbow under inlet velocity u = 20 m/s and 30 m/s , turbulent dissipation rate through the channel with double elbow under inlet velocity u = 20 m/s and 30 m/s, turbulent viscosity through the channel with double elbow under inlet velocity u = 20 m/s and 30 m/s , and the vortices through the channel with double elbow under inlet velocity u = 20 m/s and 30 m/s.

After feeding the main program by the mentioned data. Numerical treatments with the system of equations are carried out. The numerical method used her is finite element method for solution the differential equations. Figs (4) and (10) it is clear that the minimum value of flow velocity appear near the walls of elbow due to sudden change in the flow direction .in Figs (5) and (11) represent the contours of equal velocities in the channel and through elbow and the line of min velocity appear near elbow walls and the max flow velocity appear after elbows.

Figs (6) and (12) give us turbulent kinetic energy through the channel where the maximum turbulence value appear at the entrance of the flow and its minimum value in near the first elbow. Figs (7) and (13) represent turbulent dissipation rate through the channel the max dissipation rate appear near the zone of low turbulence value . Figs (8) and (14) give the behavior of turbulent viscosity through the channel were big changes occurs due to variation in flow field and kinetic energy.

Figs (9) and (15) represent the behavior of the vortices through the channel where higher vortices appear near the second elbow.

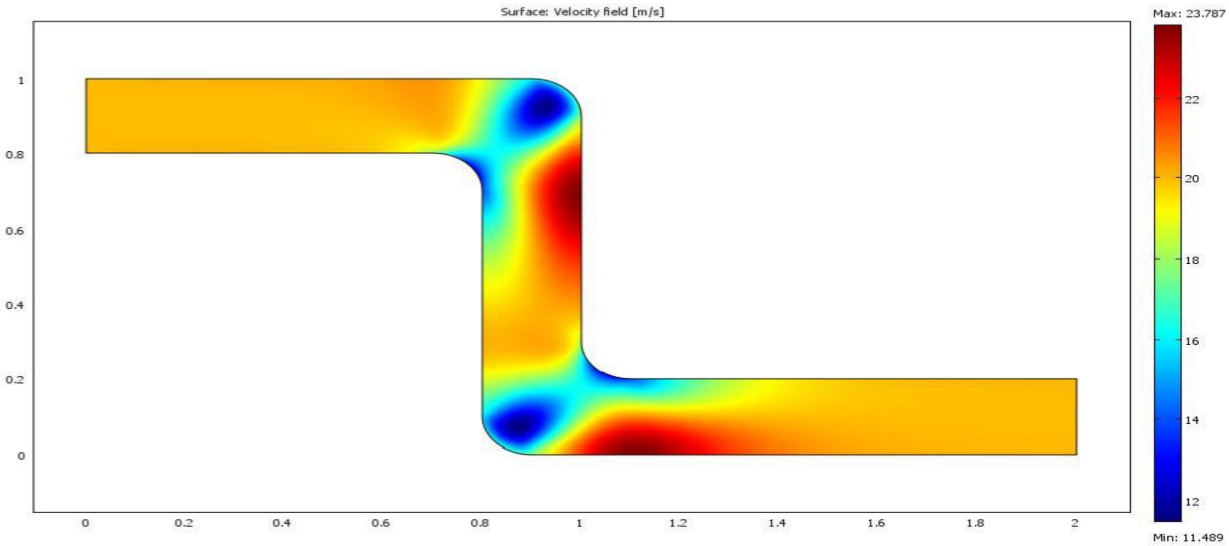

Figure (4): the velocity profile through the channel with double elbow under initial velocity u = 20 m/s

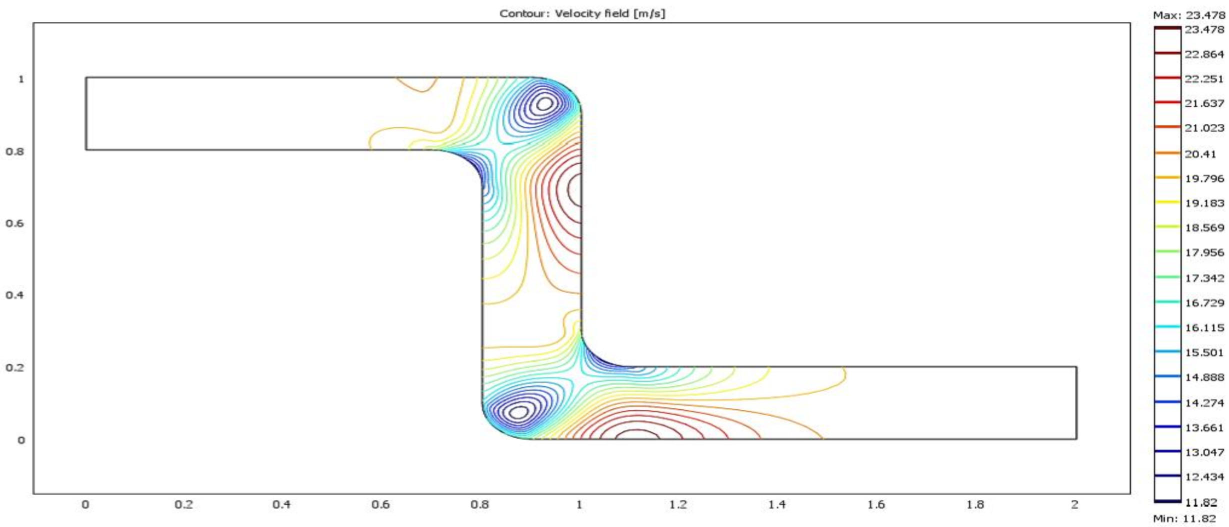

Figure (5): the velocity coontours through the channel with double elbow under inlet velocity u = 20 m/s

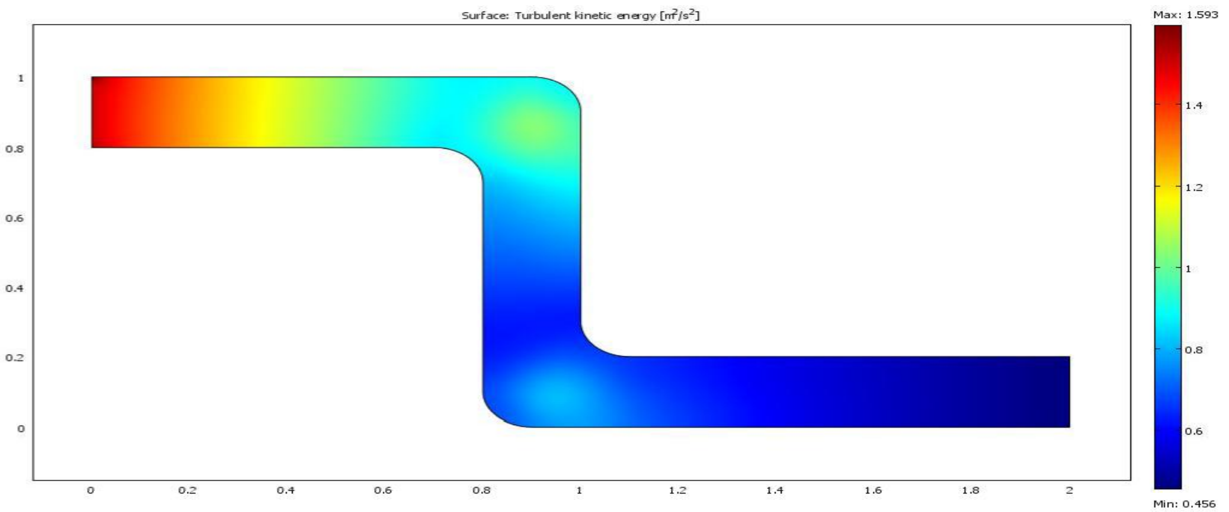

Figure (6): turbulent kinetic energy through the channel with double elbow under inlet velocity u = 20 m/s

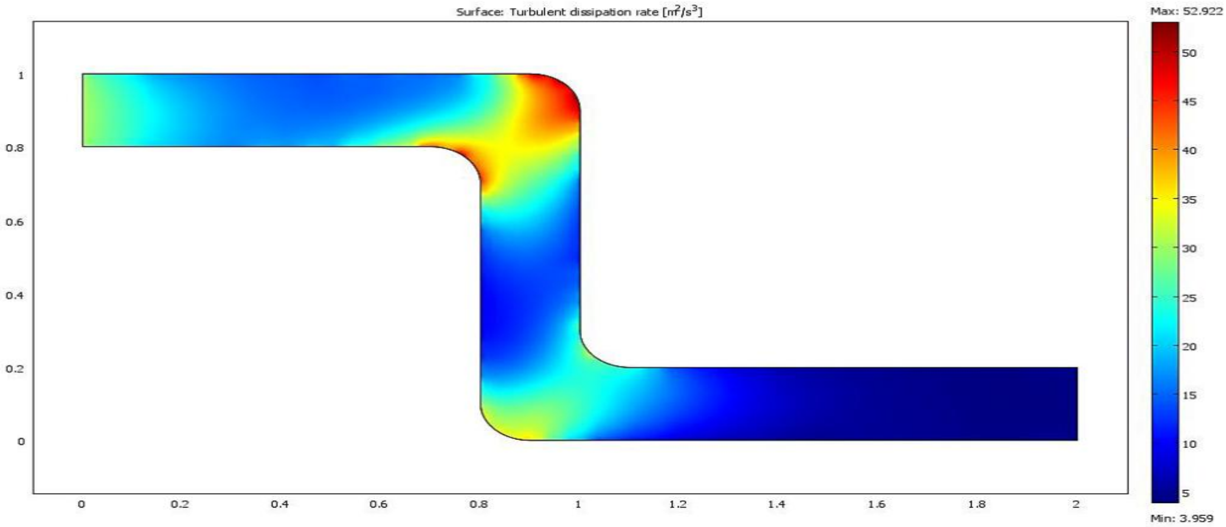

Figure (7): turbulent dissipation rate through the channel with double elbow under inlet velocity u = 20 m/s

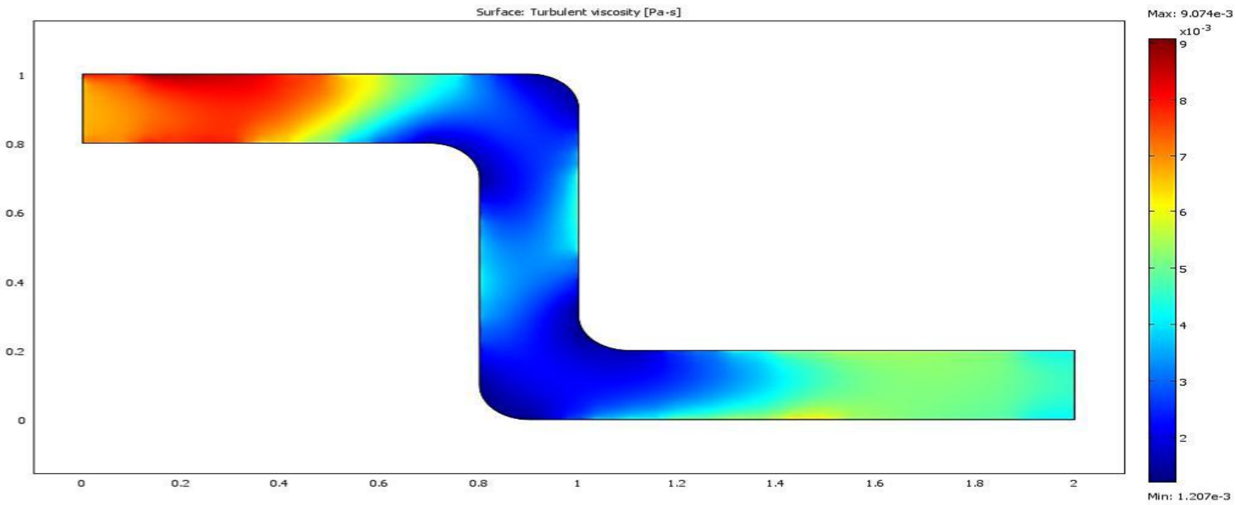

Figure (8): turbulent viscosity through the channel with double elbow under inlet velocity u = 20 m/s

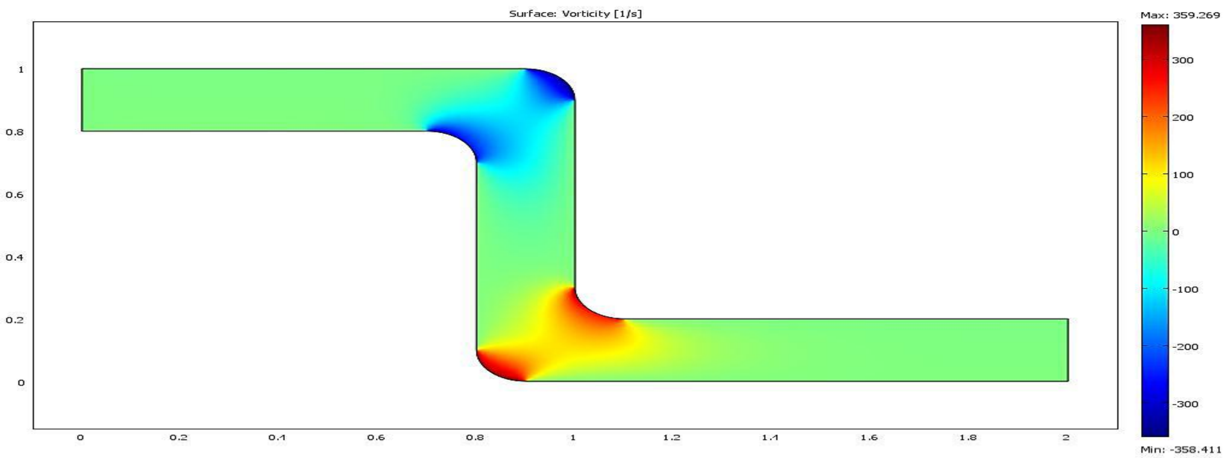

Figure (9): the vortices through the channel with double elbow under inlet velocity u = 20 m/s

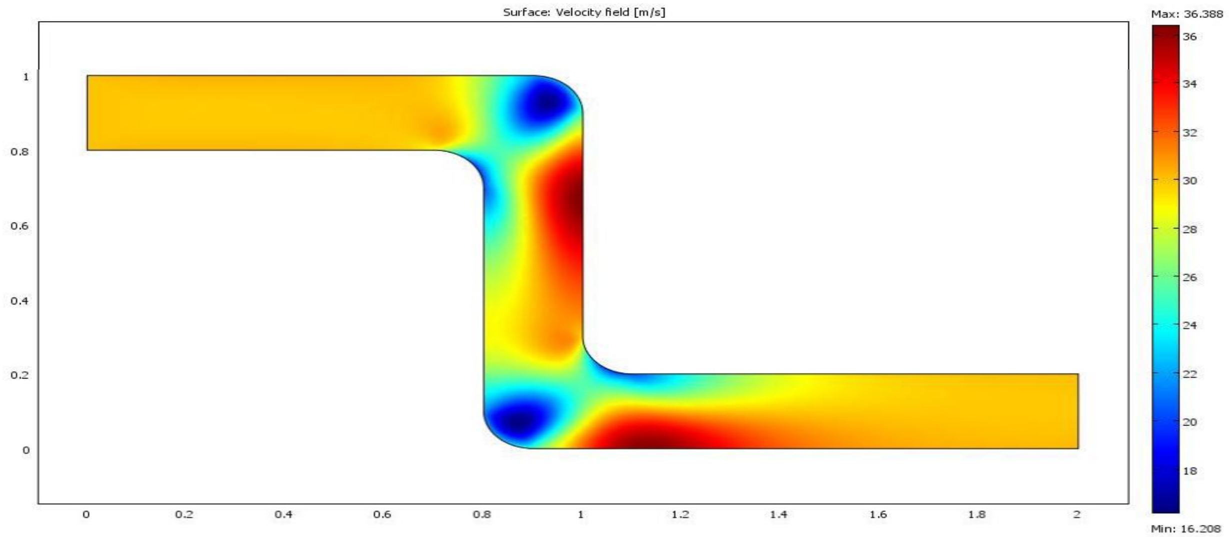

Figure (10): the velocity profile through the channel with double elbow under initial velocity u = 30 m/s

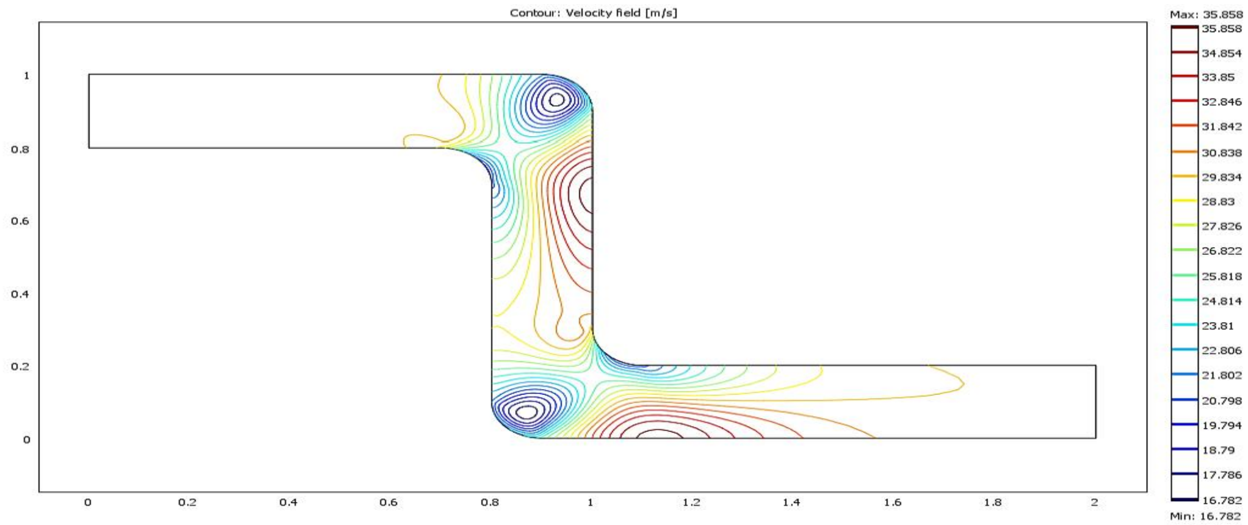

Figure (11): the velocity profile through the channel with double elbow under inlet velocity u = 30 m/s

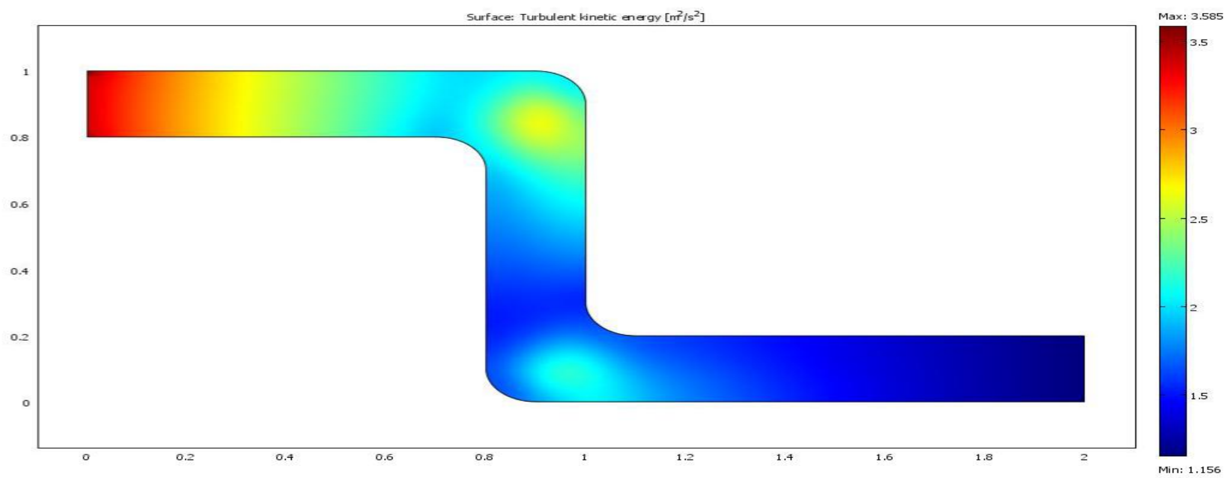

Figure (12): turbulent kinetic energy through the channel with double elbow under inlet velocity u = 30 m/s

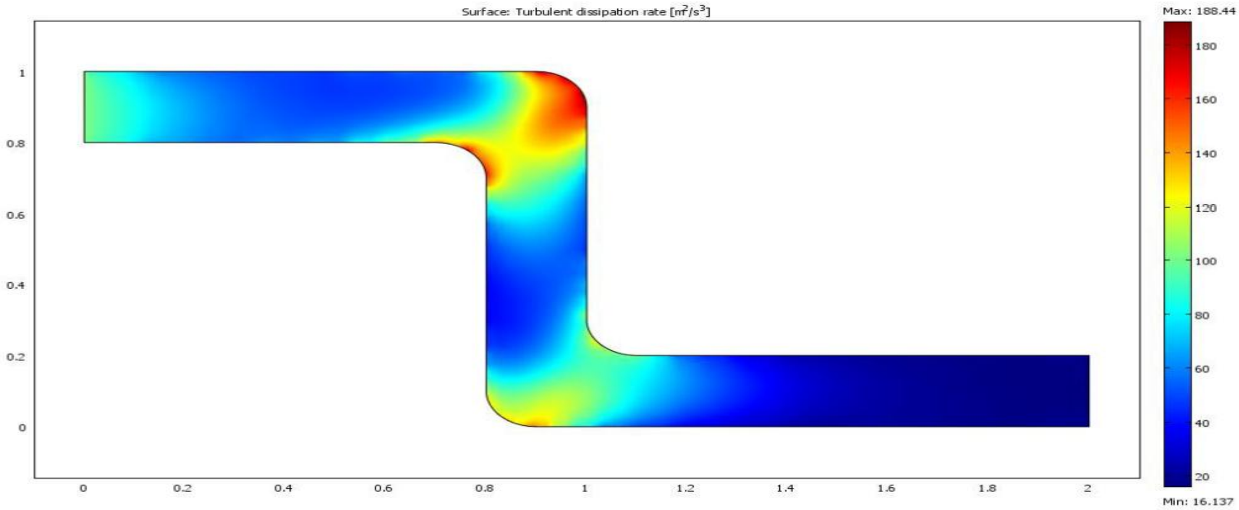

Figure (13): turbulent dissipation rate through the channel with double elbow under inlet velocity u = 30 m/s

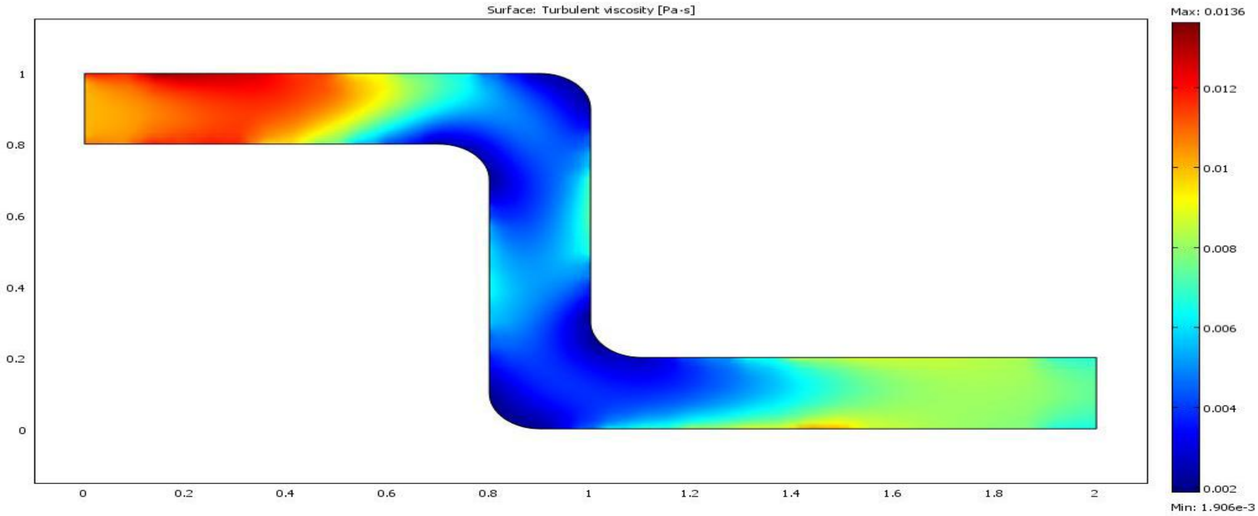

Figure (14): turbulent viscosity through the channel with double elbow under initial velocity u = 30 m/s

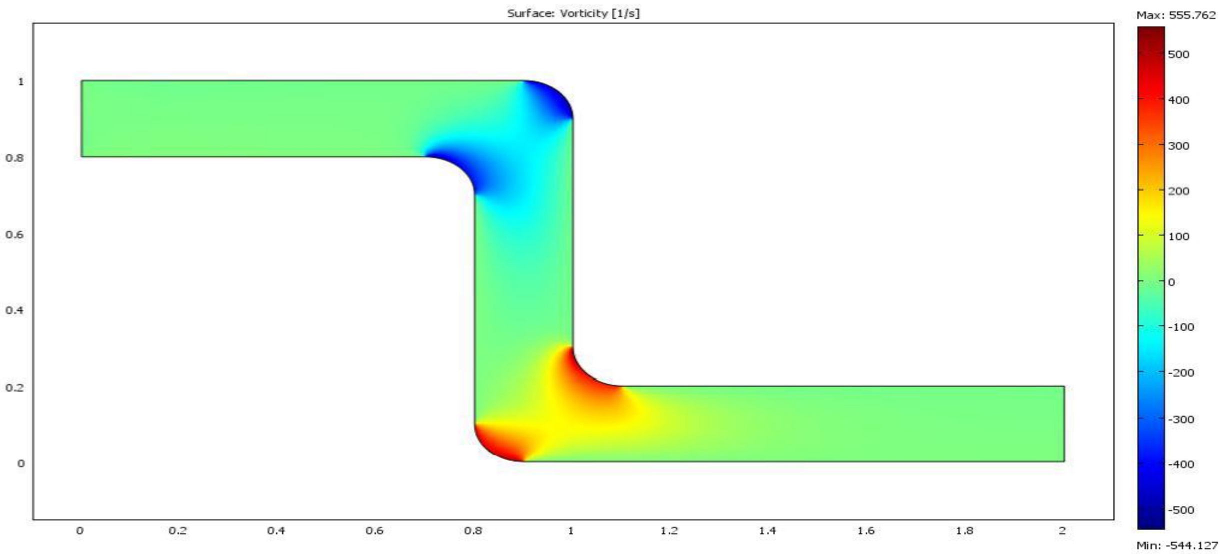

Figure (15): the vortices through the channel with double elbow under inlet velocity u = 30 m/s

Conclusions :

The present work illustrates the following conclusions

1. The finite element method can be regarded as one of the more efficient methods to analyze the flow characteristics.
2. The turbulent kinetic energy through the channel with elbow have maximum turbulence value appear at the entrance of the flow.
3. The geometry and physical assumptions play big rule for reaching stability of solution and reducing accumulative errors
4. The vortices through the channel with double elbow always concentrated in sudden change flow zone and depend on rate of turbulence

References :

1. Hawthorne, W. R., The applicability of secondary flow analyses to the solution of internal flow problems. *Fluid Mechanics of Internal Flow*, p. 263, Elsevier, 1967.
2. Horlock, J. H. and Lakshminarayana, B. Secondary flows: theory, experiment, and application in turbomachinery aerodynamics. *Annual Review of Fluid Mechanics*, Vol. 5, pp 247-280, 1973.
3. Marn, J. and Ternik, P., Laminar flow of a shear- thickening fluid in a 90^o pipe bend, *Fluid Dynamics Res.*, 38, 295-312 , 2006.
4. Edwards, M. F., Jadallah, M. S. M. and Smith, R.,.Head losses in pipe fittings at low Reynolds numbers,*Chem. Eng. Res. Dev*., 63, 43-50, 1985
5. Das, S. K., Biswas, M. N. & Mitra, A. K., Non- Newtonian liquid flow in bend, *Chem. Eng. J.*, 45,165-171. 1991.
6. Banerjee, T. K., Das, M. and Das, S. K., Non- Newtonian liquid flow through globe and gate valves, *Can. J. Chem. Engg*., 72, 207-211, 1994.
7. Bandyopadyay, T. K. and Das, S. K., Non- Newtonian pseudoplastic liquid flow through small diameter piping components, *J. Petrol. Sc. & Engg.*, 2007.
8. Das, S. K., Biswas, M. N. and Mitra, A. K., Pressure losses in Two-phase Gas-non-Newtonian liquid flow in horizontal tube, *J. Pipelines*, 7, 307-325, 1989.
9. Das, S. K., Biswas, M. N. & Mitra, A. K., Non- Newtonian liquid flow in bend, *Chem. Eng. J.*, 45,165-171. 1991.
10. Banerjee, T. K. and Das, S. K., Gas-non- Newtonian liquids flow through globe and gate valves, *Chem. Engg. Comm*., 167, 133-146, 1998.
11. Bandyopadyay, T. K., Banerjee T. K. and Das, S. K., Gas-non-Newtonian liquid flow through elbows, *Chem. Engg. Comm.*, 182(1), 21-33, 2000.