ERJ
Engineering Research Journal
Faculty of Engineering
Menoufia University

COMPUTATIONAL STUDY OF TURBULENT EMULSION FLOW THROUGH CURVED DIFFUSERS

Kamal A. Ibrahim*, Mohamed E. NourEldin***, Khalid M. Saqr**, Ahmed A. Hanafy**

* Faculty of Engineering, Menoufia University, Egypt.
**College of Engineering and Technology, Arab Academy for Science, Technology and Maritime Transport
*** MSc research student, College of Engineering and Technology, Arab Academy for Science, Technology and Maritime Transport

ABSTRACT

Emulsion flow is crucially important in numerous fluid mechanics and industrial applications. This paper investigates the flow structure and hydrodynamic parameters of emulsion flow in curved diffusers of rectangular cross section and different angles and curvature ratios. The work was undertaken using the general-purpose CFD package ANSYS FLUENT with proper verification and validation procedures. The CFD model utilized the $k-\varepsilon$ turbulence model to close the RANS equation and a single phase fluid model. The emulsion flow was presented in the computational model via its physical properties assuming homogenous mixing between the primary and secondary fluid phases. The effects of Reynolds number and diffuser geometry on the hydrodynamic performance of the diffuser as well as the flow structure were investigated and discussed.

تطبيقات ميكانيكا الموائع والتطبيقات الصناعية. تبحث هذه الورقة هيكل التدفق والعوامل تدفق مائع مستحلب (ماء+زيت)هو في غاية الأهمية في العديد من
الهيدروديناميكية المؤثرة على تدفق مستحلب في النواشر المنحنية مستطيلة المقطع العرضي وزوايا انحناء مختلفة ونسب انحناء. وتم إنجاز الأعمال
$k-\varepsilon$ تستخدم نموذج CFD مع إجراءات التحقق والتثبت المناسبة. نموذج ANSYS FLUENT للأغراض العامة حزمة CFDباستخدام باقة
ونموذج السائل ذو المرحلة واحدة. وتم اعتبار تدفق المستحلب في النموذج الحسابي عبر خصائصه الفيزيائية مع RANSللاضطراب لإغلاق المعادلة
افتراض خلط متجانس بين المرحلتين الابتدائية والثانوية للسوائل. وقد تم التحقيق في الآثار المترتبة على تغيير رقم رينولدز للسريان (نسبة قوة القصور
الذاتي الى قوة اللزوجة) وتغيير ابعاد وخواص النواشر على الأداء الهيدروديناميكي للنواشر فضلا عن هيكل تدفق المستحلب ومناقشة هذه الاثار.

KEYWORDS: Emulsions, Curved diffusers, CFD.

INTRODUCTION

Diffusers are used in many engineering applications such as turbo machines, wind tunnel piping systems, centrifugal pumps, gas turbines etc. to decelerate the flow or to convert the dynamic pressure (velocity head) into static pressure. Depending on application, they have been designed in many different shapes and sizes.[1] The curved diffuser is one of such design and is an essential component in many fluid handling systems. Curved diffusers are an integral component of the centrifugal pumps .The centrifugal machines achieves their effect by accelerating the fluid through an impeller into the working fluid. A curved diffuser is placed outboard the impeller exit to increase the efficiency.

Liquid emulsion is a mixture of two immiscible liquids, one of which is dispersed in the form of small droplets throughout the other. The dispersed liquid is known as the internal or discontinuous phase, whereas the dispersion medium is known as the external or continuous phase. Stable emulsion involves the presence of surfactants that inhibit coalescence of the dispersed droplets. In unstable emulsion, there is no additives (surfactant) added to the mixture, and the drop diameter distribution depends on the rate of drop break-up and coalescence. Emulsions form the basis of a wide variety of natural and manufactured materials, including foods, pharmaceuticals, biological fluids, agrochemicals, petrochemicals, cosmetics, and explosives. The desired rheology and stability of emulsion-based materials varies widely depending on their intended application. Manufacturers of emulsion-based materials must therefore have a good understanding of the factors that determine their rheology and stability in order to create products with the required physical characteristics [2]

Emulsions of crude oil and water can be encountered at many stages during drilling, producing, transporting and processing of crude oils and in many locations such as in hydrocarbon reservoirs, well bores, surface facilities, transportation systems and refineries. A good knowledge of petroleum emulsions is necessary for controlling and improving processes at all stages.

In a curved diffuser, due to the presence of centerline curvature, fluid near the flow axis is acted upon by a larger centrifugal force than the fluid near the walls. This centrifugal pressure difference (transverse pressure gradient) forces the faster moving fluid to move outwards pushing the fluid in the boundary layer at the outer wall around the sides towards the inner wall; thus a significant secondary flow (normal to the primary flow direction) is produced. The curved diffuser represents one form of the radial diffuser cascade, which is more complex in design and analysis than the other diffusers and the internal flow field in curved diffuser is still incomplete and needs more investigations. It is unlikely that the designer will be fortunate to find an appropriate diffuser chart to assist him in finding a near optimized curved diffuser design due to the infinite variation in configurations of curved diffuser.[3]

PHYSICAL MODEL AND GRID GENERATION

The schematic drawing dimensions shown in fig. 1 represents the computational domain of solution that deals with the flow through a rectangular cross-section area curved diffuser.

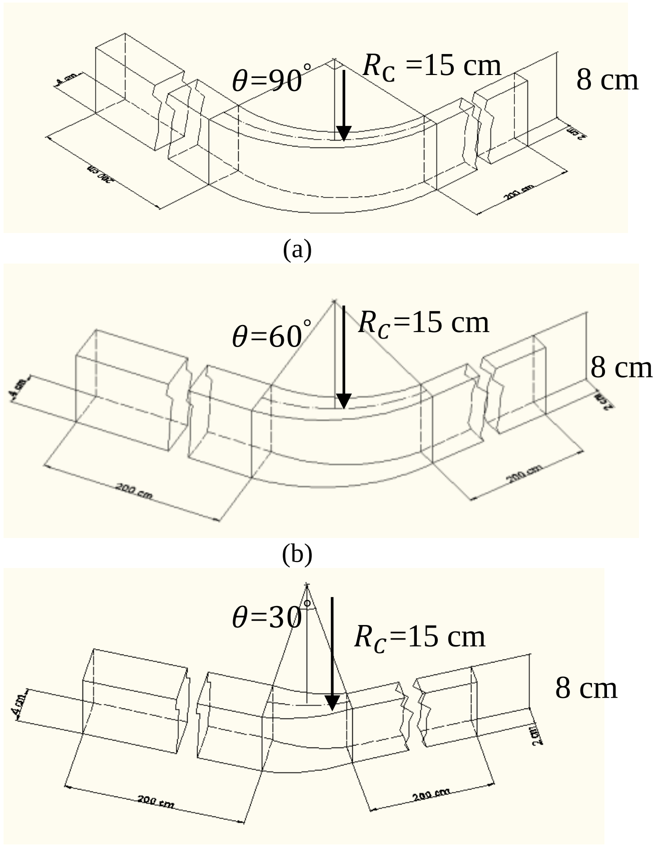


Fig 1. The schematics coordinates system for 3-D curved diffuser for angles of (a) 90°, (b) 60° and (c) 30°

The regular hexahedral grid elements are generated by the CFD software, Auto Cad 2010 is used for drawing the 3-D geometry, Gambit 2.3.16 is used for generating the mesh in the 3-D model and making the adaptation for refining the gird in both axial and radial directions as shown in fig. 2, Gambit is also used for defining the geometry walls and both the inlet and outlet of the geometry, FLUENT 14.0 is then used for the case study setup and for finding the solution.

Calculating all the energy loss through all the pipeline components is very essential in order to determine the required pump size to move the emulsion.

Many researches have been published in recent years about the flow of two phase gas/liquid mixtures through pipe fittings with attention only to single phase flow (water or gas) in curved diffuser. However, little attention has been given to the two-component liquid/liquid emulsions even though they are of the same importance from the practical point of view.[4]

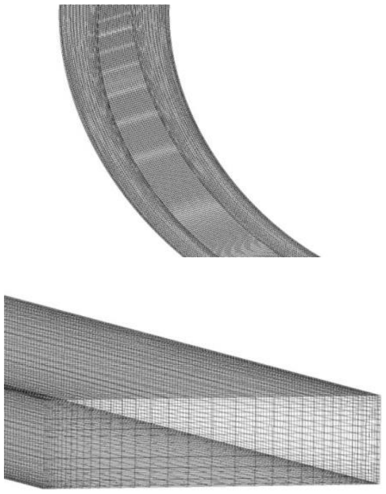


Fig 2 the computational mesh for 3-D curved diffuser

Solving a CFD problem, we're trying to approximate a continuous solution space using discrete elements. These discrete elements form the mesh. As the number of elements used to approximate the continuous solution increases, the accuracy of the CFD method used to calculate the actual solution also increases. But, as the number of elements increases, the computational time required to get the solution also increases.

Mesh or grid independence study is carried to determine this optimum point where a fairly accurate solution for the problem is found at the expense of least computational resources. In other words, given a level of accuracy (the deviation of the solution calculated from the CFD method, compared to the actual solution of the NS equations) for the solution, the mesh used is good enough to achieve that accuracy at the expense of minimum possible computational power. Using an optimum mesh, the accuracy of the results is good enough to capture all the necessary flow features, their gradients and so forth. In other words, a coarse grid will not capture all the flow features (not a solution of required accuracy) and a finer mesh will give a solution of a little higher accuracy than required but at the expense of computational power and time.

The experimental water flow measurements in curved diffuser given by El-Askary, W.A., et al [3] is predicted in the present study using five turbulence models namely, Reynolds stress model with linear pressure-strain, Reynolds stress model with quadratic pressure-strain, realizable $k-\varepsilon$, RNG $k-\varepsilon$ and standard $k-\varepsilon$ models. The comparisons show that the pressure recovery coefficient $(\mathrm{C_p})$ is successfully predicted by using the latest turbulence model (standard $k-\varepsilon$) there for it can be used as a solution turbulence model for the validation code.

MODEL VALIDATION

In order to ensure that the computational model produces physically correct results, comparison with previous experimental calculations was undertaken. The case that will be used for the current work is reported in [3].

Table-2 The case studies for the curved diffuser

Curvature Ratio	Diffuser angel			Hold up & Stability	Inlet Re			
12.5	30 deg	60 deg	90 deg	Water	25000	27000	28500	35000
				E 0.1 STABLE				
				E 0.1 UNSTABLE				
				E 0.2 STABLE				
				E 0.2 UNSTABLE				
				E 0.3 STABLE				
				E 0.3 UNSTABLE				
7.5	30 deg	60 deg	90 deg	Water	25000	27000	28500	35000
				E 0.1 STABLE				
				E 0.1 UNSTABLE				
				E 0.2 STABLE				
				E 0.2 UNSTABLE				
				E 0.3 STABLE				
				E 0.3 UNSTABLE				

The coefficient of pressure $\mathrm{C_p}$ calculated at the same locations for both the 12.5 and the 7.5 curvature ratios calculated from the experimental work of [3] water will be used in all cases at Reynolds number of 28500.The local static pressure recovery coefficient $(\mathrm{C_p})$ is used in describing curved diffuser performance and can be defined as:

$\mathrm{C_p} = \frac{(\mathrm{P_x} - \mathrm{P_{ref}})}{0.5\ \rho_\mathrm{E} \mathrm{U_{ref}^2}}$ where,$\mathrm{P_x}$ is the local wall measured static pressure and $\mathrm{P_{ref}}$ is the reference pressure and $\mathrm{U_{ref}} = \mathrm{V_{inlet}}$ is the reference mean velocity, $\mathrm{C_P}$represents the ratio between the local static pressure difference between any location (x) and reference pressure tap location (ref) to the reference dynamic pressure. The reference pressure tap location is taken at a distance 20 mm upstream of the curved diffuser entrance, fig 3 show the comparison results, the average error analysis for the same location is presented in Table-1,

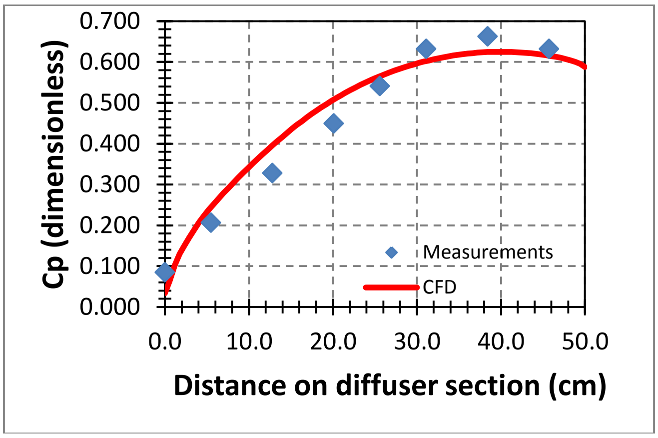


Fig 3. Validation comparison for the CFD model.

Table -1 average error analysis

Curvature ratio	Max Error	Min Error
12.5	21.6 %	0
7.5	6 %	0

The comparisons show that there are acceptable agreements between the predicted values of the present CFD code and the experimental measurements of [3], pressure recovery coefficient is successfully predicted by using the CFD code and the CFD model can be used as a solution model.

CASE STUDIES FOR CURVED DIFFUSER

Table-2 shows all the case studies for the curved diffuser, while Table-3 present the geometrical parameters of the curved- diffuser models

Table-2 The case studies for the curved diffuser

Table-3 The geometrical parameters of the curved-diffuser models (dim. in cm)

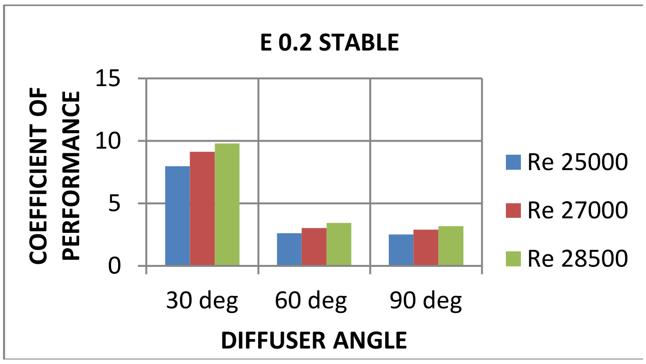
model	R_c	W	W_{exit}	B	CR	AR	θ^0
(1)	25	2	4	8	12.50	2.00	30
(2)	25	2	4	8	12.50	2.00	60
(3)	25	2	4	8	12.50	2.00	90
(4)	15	2	4	8	7.50	2.00	30
(5)	15	2	4	8	7.50	2.00	60
(6)	15	2	4	8	7.50	2.00	90

RESULTS AND DISCUSSION

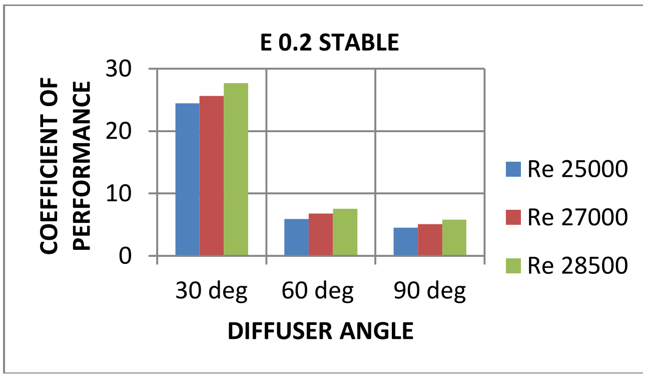
THE COEFFICIENT OF PERFORMANCE

The coefficient of performance is defined in the present study as the pressure drop per unit length through the diffuser divided by the total pressure drop per unit length through the test model, it aims to show the effect of the various diffuser parameters on the overall model, as it shows the effect of each curved diffuser at the pressure recovery of the overall model not only the diffuser section. The coefficient of performance is calculated for each model and then compared after varying the inflow and the diffuser parameters , an example of the effect of the diffuser curvature angle for different inflow Reynolds number is shown in fig 4 through while an example of the effect of the

Inflow holdup and stability is shown in fig.5



Curvature ratio 12.5



Curvature ratio 7.5

Fig 4 Inflow Reynolds number and curvature angles effect on the COP through different diffuser curvature ratios for 0.2 stable flows

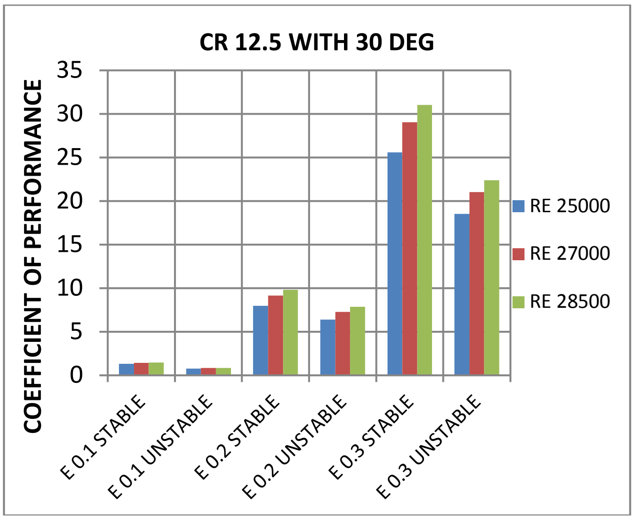


Fig 5 the effect of emulsion holdup and stability on the COP for curved diffuser with curvature ratio 12.5 and 30 degrees curvature angle

From the results it is found that the COP is proportional to the Reynolds number, and the unstable emulsions have a smaller COP compared to the stable emulsions, the CR is found to be inversely proportional to the COP.

THE DIFFUSER LOSS COEFFICIENT, (K_d)

Diffuser loss coefficient is defined as the ratio of the loss in total pressure across a given diffuser to the reference dynamic pressure at the diffuser inlet

$$K_d = \frac{p_{inlet} - p_{outlet}}{0.5\rho_E {U_{ref}}^2}$$

An example of the effect of Reynolds number on the energy-loss coefficient for all models carrying water and (stable/unstable) emulsions at different holdups are presented numerically in fig.6, while an example of the emulsion holdup effect is presented on fig 7,

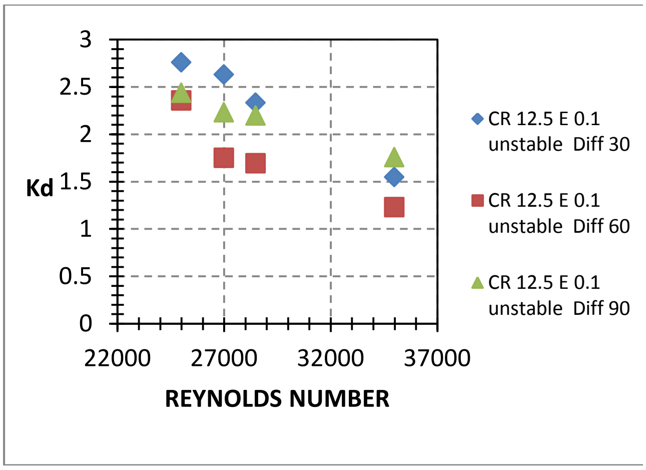
For all the models the diffuser loss coefficient K_d decreases as the inflow Reynolds number increases,

The diffuser models with curvature angle $\theta = 60°$ had the lower diffuser loss coefficient K_d at almost all the cases compared to the models with $\theta = 90°$ and $\theta = 30°$ at the same inflow parameters, the cause of this is explained while studying the separation later, It is found that the models with $\theta = 30°$ exhibits a significant separation because it is the model with the higher diffusion angle and the models with $\theta = 90°$ are with the smallest diffusion effect and are greatly influenced with the presence of the secondary flow resulting from the centrifugal force

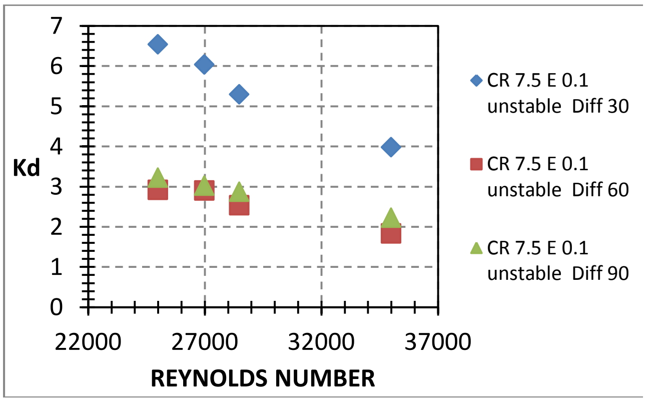
The models with $\theta = 60°$exhibit the lower values of the energy-loss coefficient compared with the other tested models, therefore the best diffuser curvature angle is $\theta = 60°$.

As the curvature ratio CR decreases the diffusion angle increases as a result of that the model with CR =7.5 and θ = 30 has the biggest diffusion angle and as a result exhibits significant separation which leads to the higher energy loss coefficient compared to the rest of the tested models.

From the results it is found that the energy-loss coefficients of the curved diffuser models are strongly affected by the emulsion holdup and stability (stable/unstable). The unstable emulsions have a lower energy-loss coefficient compared to the stable emulsions.



(a) Curvature ratio CR=12.5



(b) Curvature ratio CR=7.5

Fig.6 effect of inflow Reynolds number on the diffuser loss coefficient k_d for 0.1 unstable emulsion inflow with different diffuser curvature ratio CR and diffuser curvature angle θ

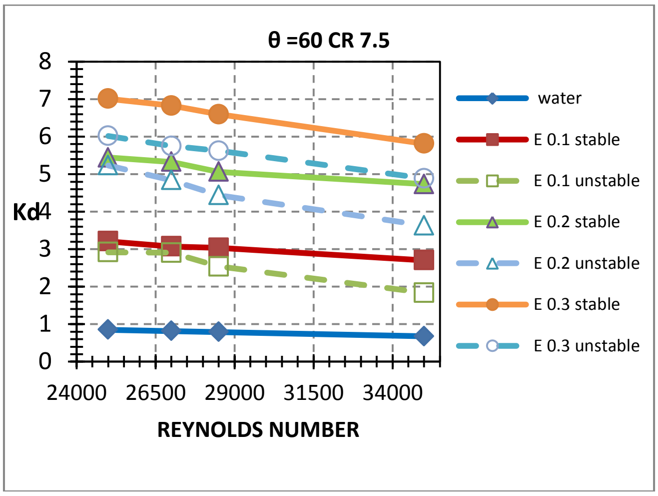


Fig.7 effect of emulsion holdup and stability on the diffuser loss coefficient k_d for curved diffuser with curvature ratio CR=7.5 and diffuser curvature angle θ = 60°

THE VELOCITY STREAM LINES

In order to study the flow regimes in accordance with the curved diffuser geometry and inflow parameters such as inflow Reynolds number and holdup ratios as well as the flow stability the stream lines is going to be presented in this section to show visually the flow behavior with variable changes.

The primary fluid mechanics problem of the diffusion process is caused by the tendency of the boundary layers to separate from the diffuser walls if the rate of diffusion is too rapid (short diffuser length). The result of too rapid diffusion is always large losses in stagnation pressure. On the other hand, if the rate of diffusion is too low, the fluid is exposed to an excessive length of wall and fluid friction losses become predominant.

Fig 8 shows an example of the effect of changing the diffuser curvature angle θ on the velocity stream lines and separations in the models with CR = 12.5, while fig 9 shows an example of the emulsion inflow Reynolds number it can be seen from these figures that the stall location is strongly affected by changing the curved diffuser geometry and its inlet flow holdup and Reynolds number.

It can be seen from the results that the models with θ = 30 exhibits significant separation and this is explained as a result of having the bigger diffusion angle, the curvature ratio is inversely proportional to the diffusion angle as a result the model with CR = 7.5 and θ = 30 have the most significant separation,

The effect of the centrifugal force causing secondary flow can also be observed from studying the velocity stream lines, as the curvature ratio increases the centrifugal force created in the curved diffuser decreases as a result the secondary flow superimposed on the mean flow decreases, which in turn causes a reduction in the curved diffuser energy loss coefficient for the different flows.

The rapid rate of diffusion caused by the short diffuser length of the models with $\theta = 30°$ causes the boundary layers to separate from the diffuser walls and also causes large losses in stagnation pressure, which explains the increase of curved diffuser energy loss coefficients of the models with $\theta = 30°$.

On the other hand the models with $\theta = 90°$ have a low rate of diffusion compared with the models with $\theta = 60°$ but due to the excessive length of the models with $\theta = 90°$ the fluid friction loses became dominant which explains why the energy loss coefficients of the models with $\theta = 60°$ is lower than the energy loss coefficients of the models with $\theta = 90°$.

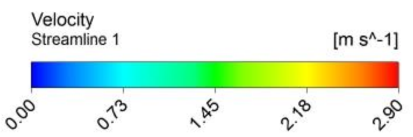
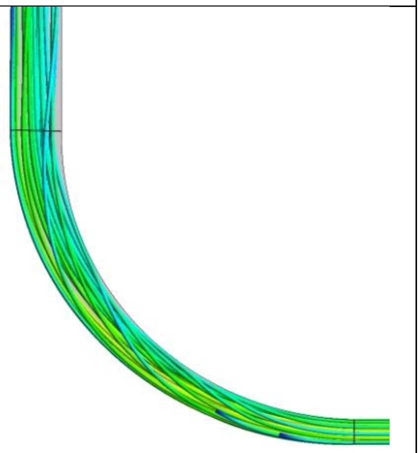
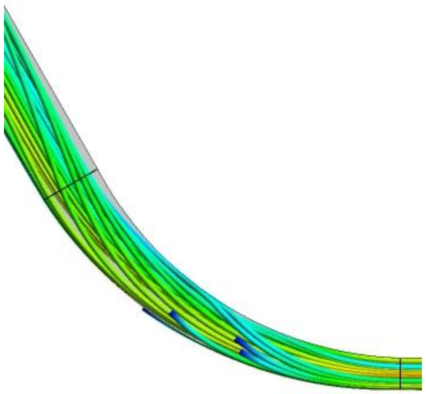
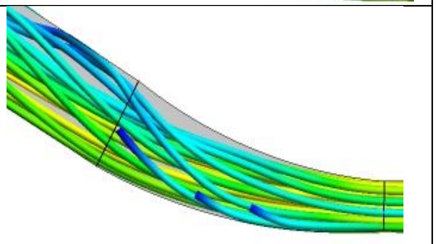
0.1 stable CR 12.5	
Re 25000 $\theta = 90^{\circ}$	
Re 25000 $\theta = 60^{\circ}$	
Re 25000 $\theta = 30^{\circ}$	

Fig 8. θ effect on velocity stream lines

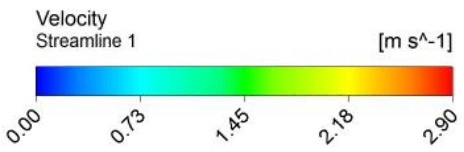
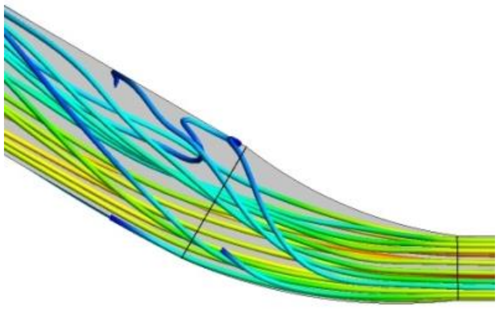
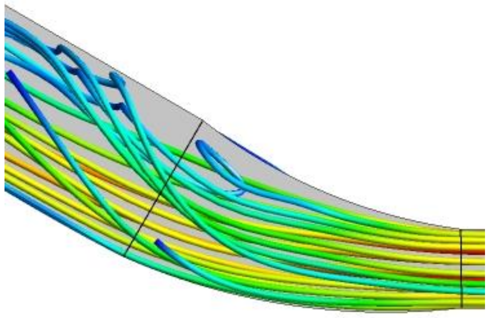
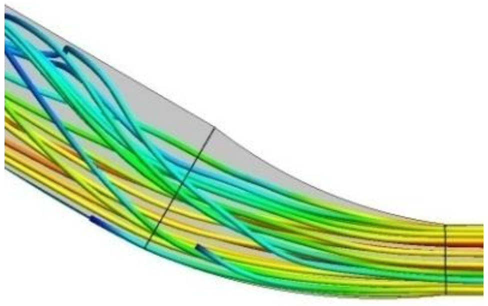
0.1 stable CR 7.5 $\theta = 30^{\circ}$	
Re 25000	
Re 27000	
Re 28500	

Fig 9. R_e effect on velocity stream lines

CONCLUSION

Based on the numerical results and related discussions the following conclusions can be drawn from the present study:-

1- The coefficient of performance increases as the curvature ratio decreases.
2- The coefficient of performance increases as the Reynolds number increases.
3- The geometrical parameters (diffuser curvature angle and curvature ratio), inflow Reynolds number, emulsion status (stable or unstable) and emulsion holdup strongly affect the curved diffuser performance.
4- The unstable emulsions have a lower coefficient of performance than the stable emulsions
5- For all models the diffuser loss coefficient K_d decreases as the inflow Reynolds number increases
6- The diffuser models with curvature angle $\theta = 60°$ have the lower diffuser loss coefficient K_d at almost all the cases compared to the models with $\theta = 90°$ and $\theta = 30°$ at the same inflow parameters.
7- The separation appears clearly with the smaller diffuser curvature angles as the diffusion is more observed.
8- As curvature ratio CR increases the diffusion effect decreases and the separation decreases.
9- The unstable (o/w) emulsions exhibits lower values of the energy loss coefficient k_d compared with stable (o/w) emulsions
10- Finally the results and conclusions provided by this study should be interesting and useful to scientists and engineers who are working in chemical and petroleum industries.

REFERENCES

[1] Sullerey, R., B. Chandra, and V. Muralidhar, Performance comparison of straight and curved diffusers. Defence Science Journal, 2014. 33(3): p. 195-203.

[2] Chanamai, R. and D.J. McClements, Dependence of creaming and rheology of monodisperse oil-in-water emulsions on droplet size and concentration. Colloids and Surfaces A: Physicochemical and Engineering Aspects, 2000. 172(1): p. 79-86.

[3] El-Askary, W.A., et al., Emulsion (Oil-in-Water) Fluid Flow in Curved Diffuser. International Journal of Fluid Mechanics Research, 2013. 40(3).

[4] Poynter, W.G. and R. Simon, Pipelining oil/water mixtures. 1970, Google Patents.

NOMENCLATURE

Φ	Holdup
μ	Viscocity
ρ	Dinisity
R_e	Reynolds number
K_d	Diffuser energy loss coefficient
COP	Diffuser coefficient of performance
θ	Diffuser curvature angle
CR	Diffuser curvature ratio
C_p	Local static pressure recovery coefficient
R_c	Diffuser radius of curvature
p_{inlet}	Diffuser inlet pressure
p_{outlet}	Diffuser outlet pressure
U_{ref}	Diffuser inlet velocity
W	Curved diffuser inlet width
W_{exit}	Curved diffuser exit width
B	Curved diffuser height
AR	Curved diffuser area ratio
E	emulsion