ISSN 2395-1621

Investigation of performance of vane type separator for marine application by using numerical analysis

[#1]Amit Vikram Singh, [#2]Girish Barpande,[#3] Vitthal Patil

[1]singh.amit557@gmail.com

[2]girish.barpande@mitpune.edu.in

[3]vitthal.patil@freudenberg-filter.com

[#1]Department of Mechanical Engineering, Maharashtra Institute of Technology, SPPU, pune 411038, (M.H.), India

[#3]Freudenberg Filteration Technology,pune(M.H.), India

ABSTRACT

The gas-liquid droplet separator is industrial equipment that has been widely used for more than a century. Due to its industrial relevance, understanding and predicting the performance of droplet separator in terms of pressure loss and collection efficiency is important. The currently used mathematical models for the prediction of droplet separator Performance exhibit limited accuracy. An alternative approach is to simulate the gas-droplet flow field in a droplet separator profiles by computational fluid dynamics (CFD). As a result of the recent progress of computational power and numerical techniques, CFD has been widely applied to industrial flow problems. There are various equipments used for liquid gas separation for example cyclones, vane type separators, packed bed separators, scrubbers etc. As the main focus is to reduce pressure drop and increase collection efficiency (removing micro size droplets from air stream) vane-type separator are promising alternative. They have comparatively low pressure drop capability and Liquid load capacity of vane type separator is also high. Due to these reasons these separators are the choice of marine applications. In marine application the main challenge is that sea water comes out with salt so design of separator is in such a way that it will restrict both salt and water. When water will pass through vane profiles then the water droplet will be trapped by drainage channels and due to inertial forces and the gravity, Water droplet will fall into drainage section and only stream of air will flow to the outlet section with some pressure drop.In the presented work an attempt is being made to predict the collection efficiency and pressure drop value for a vane type separator geometry by using numerical analysis. The solution is obtained on 2 dimensional grids for a single vane. A commercial code STARCCM+ with the Lagarangian multiphase flow physics was used to obtain various properties of flow through the separator. Values of pressure drop and droplet collection efficiency is obtained for various flow rates and droplet size. These values are compared with the available experimental results of the actual full 3D vane type separator and comments are made on utility of the numerical simulation for the design of the separators

ARTICLE INFO

Article History

Received :18th November 2015

Received in revised form :

19th November 2015

Accepted : 21st November , 2015

Published online :

22nd November 2015

Keywords— **Computational Fluid Dynamics, Flow Rate, Vane Demister, Pressure Drop, Collection Efficiency, Lagrangian Multiphase**

I. I INTRODUCTION

It is sometimes vital to remove droplets from gas streams as this liquid drops could cause damage to downstream equipment. The offshore activities in the oil- and gas industry have grown strongly in the last fifteen years. Droplet separators are devices that can accomplish this job effectively. Choice of selection from these devices depends on the composition of the mist or droplet, size, flow rate, temperature, and other operating conditions.

The GT (gas turbine) compressors used in marine application are involved with gas streams are susceptible to severe damages if liquid content of the gas flows is not mainly removed. GT compressors operate in high humidity areas (off shore) require droplet separators to remove entrained liquid in the air to avoid soaking of the air filters at their intakes. The droplet separators are designed in such a way that humid air will have to travel through angled passages so that water droplets can be trapped by trapper, usually by inertial impingement and drained out. A good design is necessary to ensure efficient trapping of water droplets without compromising the pressure drop.

The performance of droplet separators demisters) depends on many variables such as orientation of the device, flow speed, material of construction and design of the passages. To assess the performance of droplet separators, two variables are taken for close look, which are the droplet collection efficiency in term of percentage of accumulated liquid and the pressure

drop penalty. Both objective pressure drop and collection efficiency will be predicted by CFD simulation.

Vane mist eliminators or demisters find wide applications in marine application for downstream equipments since they do not clog easily and offer lesser resistance to flow than other types. As vanes are less susceptible to re-entrainment and flooding, these are more effective at higher velocities and handle higher droplet sizes while the wire mesh demisters are more suitable for removing smaller particles at lower velocities. Vane type demisters have high vapor handling capacity than wire mesh types which can be utilized in chambers operating at wide range of pressure and temperature. Hence the size of the flashing equipment can be reduced considerably. Wave-plate mist eliminators generally consist of parallel blades or vanes spaced to provide passage for vapor flow and profile with angles to provide sufficient change of direction for liquid droplets to impact, coalesce and drain from the surfaces of the plates. The wave or vane mist eliminators are typically designed for the gas to enter a narrow path in between the appropriately shaped plates and the flow has to follow the plate geometry. Normally, when the vapor and entrained liquid droplets pass through a demister, the vapor moves freely through the demister pad but the liquid droplets, due to their greater inertia, cannot make the necessary sharp turns. As a result the droplets are thrown into contact with the plate surfaces and briefly held thereby impinging on the plates. As more droplets enter the pad and collect on the surfaces, they grow in size, run down to the bottom surface of the plate separator and drain and fall from the unit. Thus the entrained liquid is removed from the gas flow effectively, usually by inertial impingement.

The air flow carries the droplet but the effect of droplets on the airflow is negligible. The liquid droplet will be separated when the gravity force is larger than the drag force of the up flowing gas stream. Drainage slots on vane surface also help to allow liquid to disengage from gas stream. Few of these drops when re-entrained into the stream at higher vapor velocities, lead to reduced droplet collection efficiency. To overcome this problem, hooks are introduced at locations where the deposited liquid may accumulate and droplet re-entrainment is reduced or prevented.

The drainage hooks help to capture the droplets and air velocity is higher at the cross section where hook is affixed, the separator shows high efficiency even though the inlet velocity is low. The presence of the hook promotes the formation of eddies which separates the fine droplets, but also result in greater pressure drop due to their strong turbulence dissipation. Water droplet separation efficiency and pressure drop across the demister are the primary parameters of interest in a desalination process, to assess the performance of the demisters. Measuring droplet distribution and number of droplets before and after the demister vanes to assess its droplet separation efficiency using experiments is costlier and also needs more skill and expertise. In order to achieve a tradeoff between the separation efficiency and pressure drop for different vane demister geometries, and to choose an optimal combination of parameters, numerical simulation studies need to be carried out. Several computational studies have been carried out by several researchers in the past, to find the influence of various geometrical parameters and numerical choices on the performance prediction of flow through straight vane demisters.

Wang et al. [1] conducted Computational Fluid Dynamics (CFD) studies using a commercial CFD code to carry out a comprehensive numerical investigation on droplet removal efficiency and pressure drop of straight vane demister by varying the parameters like inlet gas velocity, bend angle and rear pockets. Standard k–ε turbulence model was used to simulate the gaseous phase and turbulent dispersion effects on droplet trajectories were not taken into account. However, no comparison with experimental data was reported. Galletti et al. [2] performed a numerical simulation for gas flow and droplet motion in a straight vane demister with drainage channels using Shear Stress Transport (SST) turbulence model and compared the result of their simulation with the experimental results of Ghetti [3]. The results indicated that the SST turbulence model gives better results than standard k–ε model.

James et al. [4] conducted numerical simulations of the gas flow and liquid droplet dispersion, coupled with mathematical models of the film deposition and separation. Commercial CFD software CFX with standard k–ε model was used to predict the primary turbulent flow field and a separate eddy-interaction model was then used to predict the liquid droplet dispersion and deposition for a variety of droplet size distributions at inlet to the eliminator. They attempted to provide a method of determining whether, under a given liquid loading, re-entrainment takes place.

Jia et al. [5] studied the droplet behavior in the wave-type flow channels,to understand the breakup of droplets by impingement on liquid film to form secondary droplets. Realizable k–ε model was applied with standard wall function for wall node treatment, when the hydraulic Reynolds number was above 4000. The suggested numerical method showed the pressure drop and separation efficiency in good agreement with the experimental data. But secondary droplet generation from droplet breakup was not discussed in detail.

Rafee et al. [6] studied the droplet transport and deposition in the turbulent airflow inside a wave-plate mist eliminator, using an Eulerian–Lagrangian computational method. The

Reynolds Stress Transport Model (RSTM) with standard wall functions and with enhanced wall treatment was used for simulating the airflow field using a code developed in-house. Their results showed that the enhanced wall treatment improved the predictions of the droplet removal efficiency especially for small droplets. On the other hand, the RSTM with standard wall functions cannot predict the removal efficiency correctly, especially for low gas velocities.

Josang et al. [7] made an experimental investigation on a curved vane demister to measure the air velocities in strategic locations. Rafee et al. [8] carried out numerical simulations based on Eulerian– Lagrangian method and compared their results with the experimental data of Josang et al. [7],where the turbulent droplet laden air flow inside a single passage of a curved vane demister is reported. Their results show that by including the wall reflection terms in transport equations of the Reynolds stresses, better predictions can be achieved than those obtained by RSTM without wall reflection terms. The distribution of water droplets, its generation and deposition mechanisms using the Phase Doppler Anemometry (PDA) technique is discussed in detail by Josang [9]. A brief summary of droplet generation mechanisms including droplet–droplet interaction, droplet break up, splashing of impinging droplet and re-entrainment from liquid film is reported. Numerical simulation of air flow carrying water droplets through the curved demisters is also carried out and the results are compared with the experimental data.

Galletti et al. (2008) studied the importance of eddy interaction models by simulating two vane demisters. The results of their study indicated that the EIM model was unable to predict realistic results for a range of droplet sizes. Comparisons of the results with some experimental data are reported in their paper. They suggested a modified EIM model as well as a model for turbulent flow in low velocities.

Recently, Narimani and Shahhosseini (2011) stated the importance of vane geometry on removal efficiency and pressure drop using CFD. In their study the effect of increasing gas velocity on vane demister separation efficiency has been analyzed. Among re-entrainment mechanisms, it was supposed that breakup of the droplets by their impingement on a liquid film and re-entrainment from the liquid film is likely to occur. They optimized vane geometries using response surface methodology. Their model was validated using published experimental data.

Having evaluated the results of the literature it indicated that the experimental and numerical studies with droplet transport and deposition reported are mostly on wave type or straight vane type demisters and the research work on curved vane demisters is found to be very little. The numerical studies reported so far did not give a clear recommendation of a particular turbulence model and a choice of droplet breakup models for the prediction of multiphase flow through curved vane demister. Hence a numerical study is conducted to estimate the performance of a curved vane demister, using chosen turbulence models identified from literature and from the previous work by the present authors [10] .

The results from the present computations of existing vane geometry were compared with the experimental data which is obtained by FREUDENBERG FILTERATION TECH. The primary objective of the present work is to investigate droplet collection efficiency and pressure drop of existing geometry given by the FREUDENBERG FILTERATION TECH. by applying suitable physics model options. Hence the boundary conditions are given to geometry to obtain above objectivs and the computational results are validated with the experimental results. The present study is focused on the multi-phase flow of air carrying water droplets to study the droplet entrainment, carryover and flooding in the curved vane demister and a future work is planned to include droplet evaporation and condensation.

II. PHYSICAL AND COMPUTATIONAL MODEL

A. Physical geometry of the vane separator

To investigate droplet collection efficiency and pressure drop with influence of chosen turbulence models and lagrangian model in the CFD solver i.e. starccm+ and to validate the present computational methodology, the experimental dataset of Freudenberg filteration technology on curved vane separator with drainage hooks geometry is chosen. The demister geometry on which the experiments were conducted is shown in Fig. 1 whose geometry profile details are available. The vane used in the experiments has an axial length of 500 mm in the x-direction, a transverse pitch of 30 mm in the y-direction and a depth of 150 mm in the z-direction. The ratio of vane depth in the z-direction to the vane spacing in the transverse direction is found to be 5. This indicates that the two dimensional approximation of the computational domain is valid. This assumption is in-line with Rafee et al. [8] where it was mentioned that if the depth of vane separator is much larger than the other two dimensions, the flow can assumed to be two-dimensional that's why by using cfd solver starccm+ 3D meshed model will be converted into 2D model. In fig. 2 different images of vane frame will be captured by company are shown .The pressure drop obtained by simulation are compared with the experiments data for different volume flow rate for air flow. Construction detail of proposed vane drolet geometry given by Freudenberg filteration technology is as shown in table no.

Table I

Sr No	Parameter	Description
1	Profile Model	T-200
2	Make of Profile	Freudenberg
3	Size of front Frame	592 mm x 592 mm
4	Overall Depth	182mm
5	Material of Vanes profile	Polyvinyl chloride
6	Depth of Profile	150mm
7	Pitch	30mm
8	Total length of profile	8.5m
9	Material of casing & mounting profiles	SS304

10	**Drain**	**1” Drain plug from bottom**
11	**Opening area available for air flow**	**0.22 Sq. m**
12	**Approximate Weight**	**18 Kg**
13	**Air Velocity Range**	**1.8 m/s to 4 m/s**

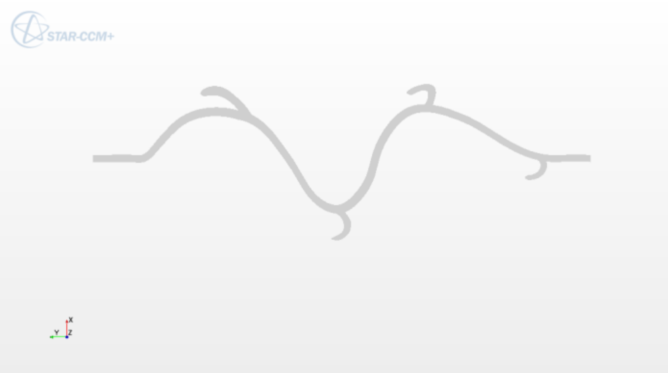
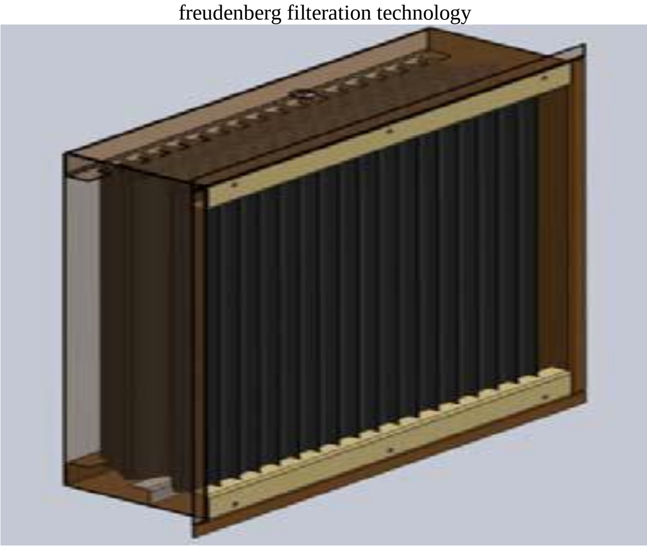
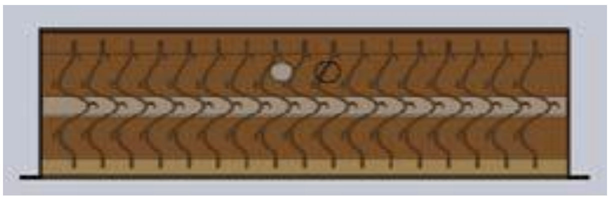


Fig. 1. Single Curved vane separator experimentally tested by freudenberg filteration technology





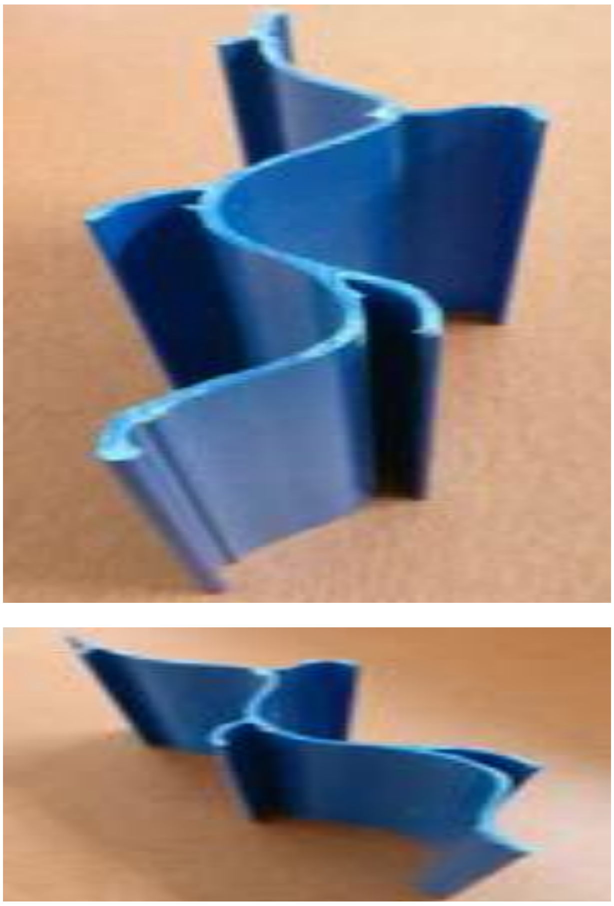


Fig. 2 images of droplet separator filter frame and vane profile

B. Computational domain

As an initial step toward using computational method to study fluid flow in the vane, the 3D geometry was structurally meshed (Fig. 3 & 4) then Meshed 3d geometry will be converted into 2d for furher simulation so that the calculation domain was divided into a finite number of control volumes (about 2068496 cells for 3D geom.) and (14551 cells for 2D geometry).because of less cells in 2D geometry as compared to 3D geometry we will use 2D geometry for simulation. Since the majority of separated droplets are expected to form a liquid film on the walls of the vane in comparison to the embedded channels, finer grids were generated near the wall region, to achieve precise results. In order to conserve computational time and still adequately provide accurate results, grid sensitivity studies were conducted to choose final grids that gave grid-independent numerical results upon further grid refinement. The dispersed and continuous phases were water droplets and air respectively.

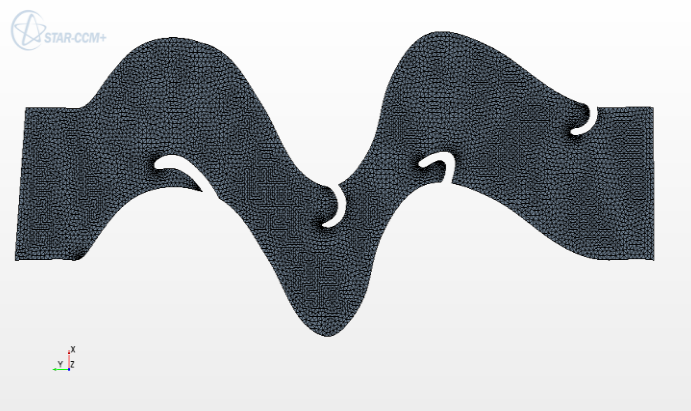


Fig. 3 volume meshed 3D geometry of vane demister

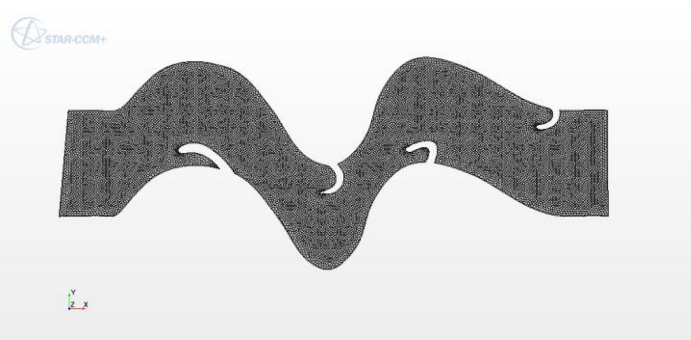


Fig. 4 volume meshed 2D geometry of vane demister

C. simulation assumptions

- Since in practice the depth of the vane is much larger than the other two dimensions, the flow is assumed to be two dimensional. Thus the two dimensional results are compared with three dimensional experimental results to judge the appropriateness of the assumption.
- Since the value of the Weber number in this study was below the critical value, film breakup was neglected.
- The droplet–film interaction at the walls is negligible.
- Re-entrainment was also not taken into account.
- The walls of the vane are assumed to be stationary and to have no slip shear conditions. The temperature of the walls is the same as the flow thus no heat transfer is happening.

Governing equations of the developed model are solved by finite volume method employing Semi Implicit Method for Pressure Linked Equations (SIMPLE) algorithm that is developed for multiphase flow using Partial Elimination Algorithm (PEA). To achieve a high spatial accuracy the second order upwind scheme was used to discrete the equations. The conservation equations were integrated in space and time. The sets of algebraic equations were solved iteratively using an explicit method. The procedure was separated as two major iteration routines. The first one was for continuous phase (gas phase) flow calculation, and the second one was for the liquid droplet flow calculation.

D. Governing equations

The governing equations used in the present computations are the continuity, momentum and turbulence equations. The following assumptions were made for the present study. A single phase fluid (air) was considered as the working fluid on which the water droplets were seeded. The computations are carried out as a steady, incompressible, two-dimensional Reynolds Averaged Navier–Stokes (RANS) solution. The conservative form of the conservation of mass or continuity equation and momentum conservation equations or Navier–Stokes equations is shown in Eqs. (1) and (2):

$$\rho \quad \nabla.(\overrightarrow{V}) = S \qquad ......(1)$$

Where $\vec{V}$ is the vector form of fluid velocity in the coordinate x and y directions. The source term S_m is the mass added to the continuous phase from the dispersed secondary phase. For the present case the right hand side of the equation is zero, since the evaporation of droplets is not considered presently

$$\rho \quad \nabla.(\overline{\,})= - \quad \nabla p + \nabla .(\underset{T}{\Rightarrow}) + \rho\vec{g} + \vec{f} \;. \qquad ....(2)$$

Where p is the static pressure, $\underset{T}{\Rightarrow}$ is the stress tensor and $\rho\vec{g}$ and $\vec{f}$ are the gravitational body force and external body forces that arise from the interaction with the dispersed phase respectively. The stress tensor is given by

$$\underset{T}{\Rightarrow} = \mu[(\nabla\vec{V} + \nabla\vec{V}^{T}) - \tfrac{2}{3}\nabla .\vec{V}I\,] \qquad ....(3)$$

Where μ is the molecular viscosity, I is the unit tensor and the second term on the right hand side is the effect of volume dilation. The resulting Reynolds stresses are modeled in the turbulence models available in the CFD code STARCCM+ making the solution a closed set. The turbulence model details and the transport equations for various turbulence models studied presently are not reported here. Interested researchers are directed to refer any standard text books in the subject area or access the help available in the solver.

The solver predicts the trajectory of a discrete phase droplet (or droplet or bubble) by integrating the force balance on the particle, which is written in a Lagrangian reference frame [11]. This force balance equates the droplet inertia with the forces acting on the particle, and can be written (for the x direction in Cartesian coordinates) as

$$\frac{du_p}{dt} = F_D\left(\vec{u} - \overrightarrow{u_p}\right) + \frac{\vec{g}(\rho_p - \rho)}{\rho_p} + \vec{F} \qquad .....(4)$$

Where $F_D\left(\vec{u} - \overrightarrow{u_p}\right)$the drag force per unit droplet mass is, $\vec{F}$ is an additional acceleration term, $\vec{u}$ is the fluid phase velocity, $\overrightarrow{u_p}$ is the droplet velocity, ρ_p is the density of the particle.

E. Numerical details

The numerical grid contains a finer near wall mesh on the separators wall surfaces, with the base size is 1 mm with surface size set which consist of target size is 75% to base size and minimum size of cell is 20%. The domain is later meshed with mesh model like polyhedral mesh, extruder, surface remesher and surface wrapper to structured mesh. The pressure drop across the demister was considered as the parameter to adjudge the mesh requirement. Computations were carried out on 2d meshed model to estimate the pressure drop for the air flow across the demister. The pressure based solver, chosen with implicit and absolute velocity formulation is used. SIMPLE algorithm is used for coupling the pressure and velocity terms. The second order upwind differencing scheme is used for convective terms and for the turbulence quantities in order to enhance the numerical accuracy of the computational results. The turbulence models tested in the present study include few of the standard options in the solver, which are predominantly reported in literature. Reynolds Stress (RSM) models recommended by [7,8] were considered for the present numerical investigations. Gravitational force ρg of Eq. (2) is considered in the present analysis as the body force acting on the computational domain, along the negative stream-wise direction.

F. Boundary conditions

The experiments were conducted at different flow rates for obtaining pressure drop and for collection efficiency for a constant flow rate by changing droplet diameter for investigate droplet collection efficiency. Air is chosen as the working medium in the computations as that of the experiments. Demister vane surfaces are declared impermeable to fluid, by specifying them as a 'wall' in the solver and set as trap for the discrete phase droplets. When this setting is used, the trajectory calculations of the droplets impinging on the wall are terminated and the fate of the droplet is recorded as "trapped". Initializing the case from the demister wall condition 'dry', the droplets are treated as adhering to the wall, making it 'wet'. It is to be mentioned that only the "trap" behavior of the walls is modeled presently and other effects like droplet shearing, re-entrainment of droplets from the walls and splashing, etc., are not considered.

The domain inlet is specified with a different mass flux and outlet is assumed to exit to open atmosphere, which is achieved by specifying a constant static pressure value of '0' Pascal at the outlet. The hook wall is chosen as "escape" boundary for the discrete phase droplets. The droplet is reported as having "escaped" when it encounters the boundary in question and the trajectory calculations are terminated. This information is sufficient for the solver to transfer information between the adjacent blade rows. The objective of the present work is to identify the solver options to predict the droplet separation efficiency of the curved vane demisters, by comparing the results with the available experimental data from Freudenberg Ltd. in the chosen domain. Further numerical analyses were not extended to cover higher inlet velocities which might be available in the commercial desalination systems. The present study will be extended in the future to cover higher velocities of water vapor (instead of air as used presently) in the curved vane demisters to understand its performance at commercial flow conditions.

G. Convergence criteria

The convergence criterion for the computational solution is determined based on scaled residuals for the equations of continuity, momentum equations and turbulence quantities specific to the respective models. The scaled residuals for solution convergence are set to 10^{-5} for all governing equations and turbulence quantities. The solution is considered to be converged when all the scaled residuals are less than or equal to this prescribed value. Computations are carried out until the steady state is reached. For few cases, the convergence is not achieved to the desired accuracy. In those cases, the iteration is continued further to a stage that the results do not vary even after 1000 iterations, thus achieving iterative convergence. For the droplets distribution estimation, sampling is done for additional 1iterations and the data summed up at the injection location and the data extraction locations were used to do the diameter statistics of the droplets.

III RESULTS AND DISCUSSION

A Pressure drop

The numerical analysis of air flow was performed in a curved vane demister to validate pressure drop and the results are discussed in the section below. Experimental values for pressure drops are depicted in fig.5 for different volume flow rate. In experimental result volume flow rate is given for entire frame of vane separator so that first volume flow rate will be converted into mass flow rate then mass flow rate will be divided by vane face area i.e. 0.22 m^2 will get value of mass flux that will be the input for air flow From table no. 1 we can say that if we increase the mass flux of air flow then pressure drop will also increase.

Table II

Volume flow Rate (m^3/h)	Mass flow Rate (kg/s)	Mass flux ($kg/m^2.s$)	Pressure drop (experimental)(pa)	Pressure drop (simulation) (pa)
493.31	0.174	0.8	6.1	5.8
997.54	0.353	1.6	25	25.6
1501.51	0.531	2.41	56.5	60
1997.68	0.707	3.21	99.8	107.5
2495.8	0.884	4.01	157	167.3
2994.95	1.061	4.82	229.6	239.6
3496.5	1.238	5.62	321.8	324.3
4010.14	1.42	6.45	425.1	421.5
5008.18	1.776	8.1	681.3	647.5

6010.3	2.13	9.7	975.7	916
6481.02	2.3	10.45	1117.6	1074.7

The CFD simulation results in this figure display less low velocity sectors in the vane with drainage channels due to more turbulence, created owing to these channels. The higher overall velocity in the channel embedded vane, meaning higher turbulence, is evident in this figure. As can be seen in Fig. 6, the gas phase is inclined to vane wall in the case of vane with drainage channels so droplet collision to wall is raised leading to higher separation efficiency. Here RSM (Reynolds stress model) is used for air flow simulation will give result which validate experimental values. By different literature reviews on vane demisters it is proved that RSM gives better result as compared to other turbulence model in vane separator. RSM give better predictions for velocity profile and pressure drop and have better agreement with result of experimental data

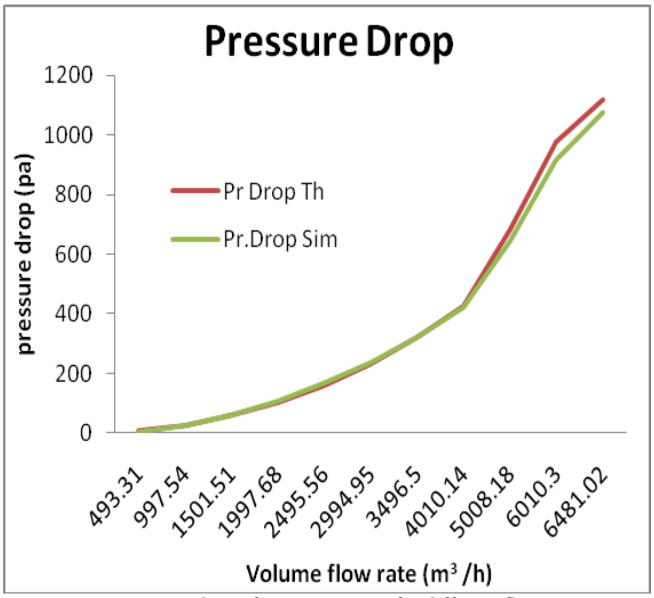


Fig. 5 pressure drop of vane geometry for different flow rates

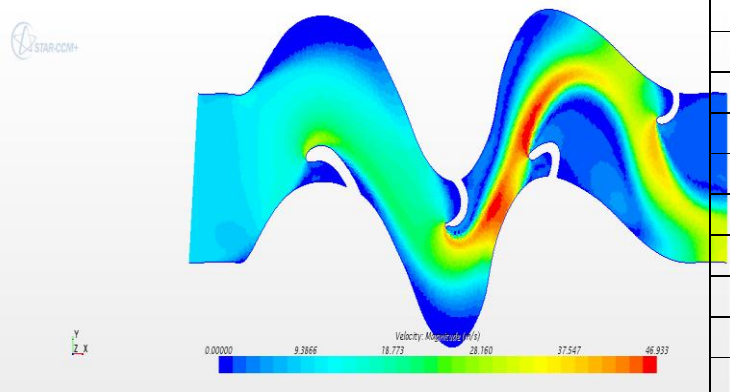


Fig . 6 velocity profile for air flow simulation

B droplet collection efficiency

The numerical droplet distribution analysis was performed in a curved vane demister and the results are discussed in the section below. The data is deduced from the solver in the form of droplet separation efficiency (η), which is given as a ratio of mass flux of droplets removed by drainage hooks to that entering the domain. In starccm+ report is made by choosing surface integral with mass flux of droplet phase which is given boundary as a wall hook and that will give total water droplet trapped and that trapped droplets mass flow rate will divided by mass flow rate at inlet.

$$\eta = \frac{\text{mass flow rate(droplet collected in vane hook)}}{\text{mass flow rate (droplet entering)}}$$

Experimental result and simulation result for different flow rates is obtained for investigating droplet collection efficiency are as follows :

Table III

I) case 1 :- flow rate 3200 m^3/h (10 % of air flow rate given to water flow rate)

Droplet diameter	Collection efficiency(experimental)	Collection efficiency (simulation)
0.24	0.01	0
0.39	0.01	0
0.59	0.03	0
0.84	-0.01	0
1.14	-0.01	0
1.44	0.04	0
1.88	0.07	0
2.57	0.06	0
3.46	0.13	0
4.69	0.16	0.02

Table IV

II) case 2 :- flow rate 4250 m^3/h (10 % of air flow rate given to water flow rate)

Droplet diameter	Collection efficiency(experimental)	Collection efficiency (simulation)
0.24	-0.04	0
0.39	-0.03	0
0.59	0.00	0
0.84	0.00	0
1.14	-0.04	0
1.44	0.02	0
1.88	0.03	0
2.57	0.10	0
3.46	0.21	0
4.69	0.33	0
6.2	0.54	0.05

Table V

III) case 3 :- flow rate 5500 m^3/h (10 % of air flow rate given to water flow rate)

Droplet diameter	Collection efficiency(experimental)	Collection efficiency (simulation)
0.24	-0.04	0
0.39	-0.05	0
0.59	0	0
0.84	0	0
1.14	-0.01	0
1.44	-0.04	0
1.88	0.04	0
2.57	0.19	0
3.46	0.26	0
4.69	0.27	0.03

Table VI

IV) case 4 :- flow rate 6900 m^3/h (10 % of air flow rate given to water flow rate)

Droplet diameter	Collection efficiency(experimental)	Collection efficiency (simulation)
0.24	-0.06	0
0.39	-0.06	0
0.59	-0.01	0
0.84	-0.07	0
1.14	-0.02	0
1.44	0.06	0
1.88	0.09	0
2.57	0.21	0
3.46	0.23	0
4.69	0.17	0.03
6.2	0.33	0.15

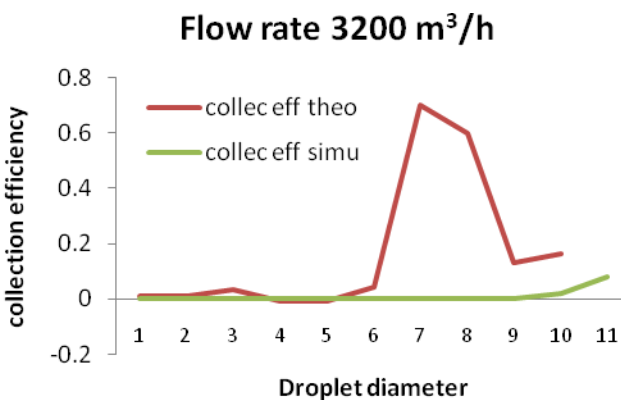


Fig. 7 (a) collection efficiency result comparison for 3200 m^3 /h air flow rate

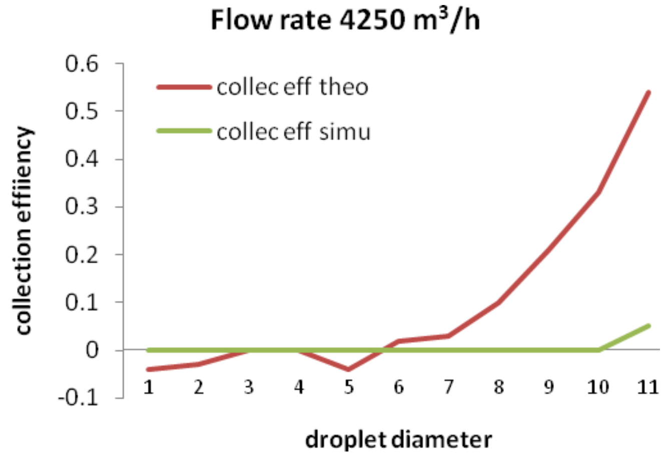


Fig. 7 (b) collection efficiency result comparison for 4250 m3 /h air flow rate

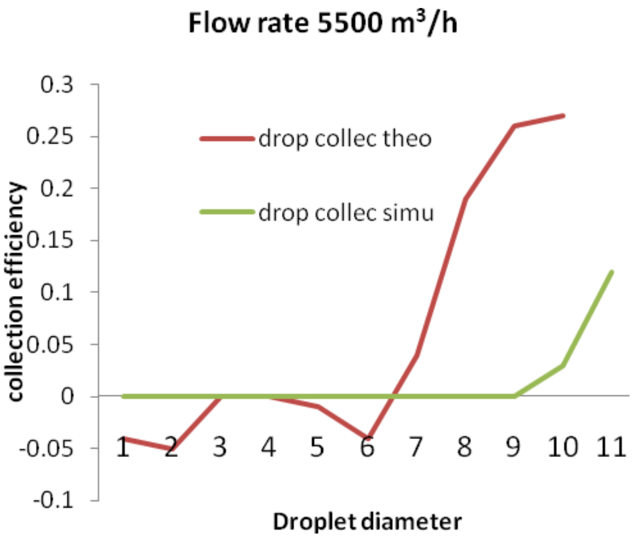


Fig. 7 (c) collection efficiency result comparison for 5500 m3 /h air flow rate

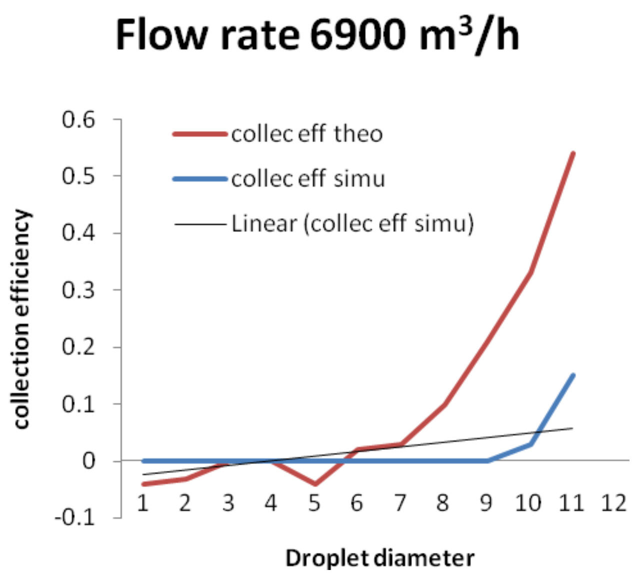


Fig. 7 (d) collection efficiency result comparison for 6900 m^3 /h air flow rate

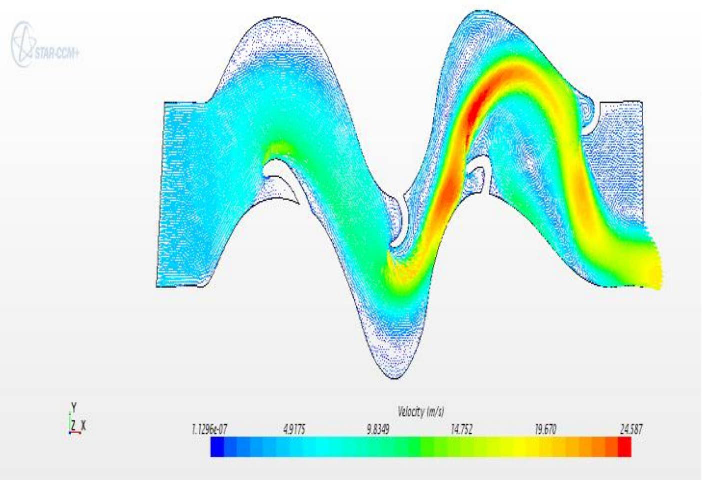


Fig 8 (a) droplet track file or trajectory of vane separator for different flow rate

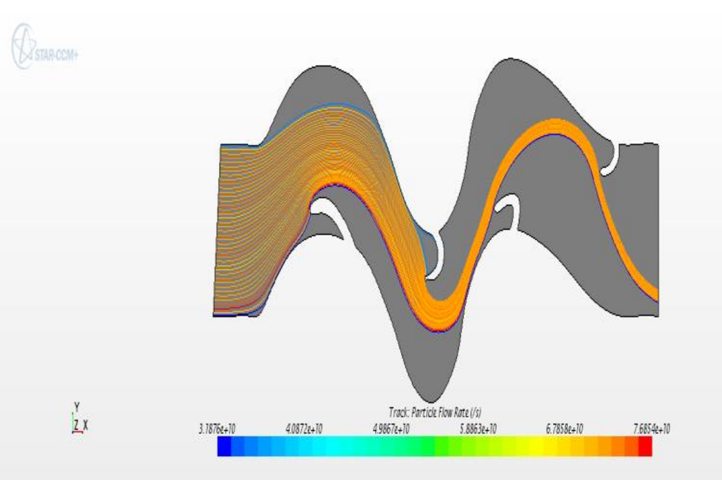


Fig 8 (b) droplet track file or trajectory of vane separator for different flow rates

In this section, the computational models with RSTM were used and the droplet transport and airflow in a mist eliminator is examined. The system studied is identical to the one studied by Phillips and Deakin (1990), where they measured the removal efficiency of a wave-plate mist eliminator experimentally. The effect of drainage channels to enhance the vane in terms of droplet trajectories is shown in Fig. 8(a) & (b). This figure illustrates that, in the case of vane with drainage channels, the droplets have a higher chance of hitting the vane walls leading to higher separation efficiency. The obtained result for droplet trajectories in Fig. 8(a) & (b) are accordance with the reported results for gas phase flow field that gas phase inclined to vane wall in the case of vane with drainage channels. This argument has been confirmed with the computation of the droplet collection efficiencies for the vane with the operating conditions (air flow rates, 10% of flow rate given to liquid droplets at inlet).

The developed computational model was applied to predict separation efficiencies of the vane using liquid droplets in the size range of 0.1–15 µm. To make comparisons between separation efficiencies for different droplet diameters, the droplets were assumed to be injected with a uniform size at the vane inlet in each simulation. The results of droplet collection efficiencies for different flow rate were compared with the corresponding experimental data of vane geometry given by Freudenberg filteration tech. in order to validate the developed computational model as shown in table III IV V VI.

The results of this computational model show an increase in separation efficiency with the rise in droplet diameter. As it can be observed, for larger droplets (diameters more than 6 µm) predictions are more consistent with the experimental data. The reason is that smaller particles have less droplet momentums due to their lighter mass and are more likely to change with the velocity fluctuation so that for small diameter droplet simulation will not give exact predictions. for all 4 cases droplet simulation will carried out by injecting as part injector of particular droplet size with water flow rate. If you are comparing all 4 flow rate droplet simulation then at particular droplet size collection efficiency will shoot up and give results before that it will shows 0% efficiency.

IV CONCLUSION

In this research, computational study has been performed to investigate the pressure drop and droplet collection efficiency of 2D vane type demisters with drainage hook geometry. Vane geometry provided by FREUDENBERG FILTERATION TECHNOLOGY Ltd. is 3D geometry. The ratio of vane depth in the z-direction to the vane spacing in the transverse direction is found to be 5. Rafee et al. [8] mentioned that if the depth of vane separator is much larger than the other two dimensions, the flow can assumed to be two-dimensional. This indicates that the two dimensional approximation of the computational domain is valid. In the selected physical model of Starccm+, the Lagrangian model approach was used to simulate droplet phase flow and RSM (Reynolds stress model) is used to simulate gas phase flow in vane type mist eliminators. The motions of liquid droplets were simulated by solving the equations of motion of the individual dispersed phase. Simulations are carried out for air flow with different flow rates and validated by the pressure drop result with experimental data. Once the primary gas flow field was calculated, the droplet motion, trajectory and deposition on the vane wall are computed by lagrangian multiphase approach. The effects of the droplets on the gas phase were also taken into account. The dispersion of the droplets due to gas phase turbulence was predicted using lagrangian approach. The CFD simulation results indicate that using vane demisters with drainage channels leads to an increase in flow turbulence as well as reduction of droplet re-entrainment. The increased turbulence obliges droplets to change trajectories and thus their impingement on to the walls rises causing the overall droplet removal efficiency to increase. By using Starccm+ solver predictions of droplet collection efficiency are carried out at different 4 cases flow rate of different droplet diameter. It can be seen that, for larger droplets (diameters more than 6 µm) predictions are more consistent with the experimental data. The reason is that smaller particles have less droplet momentums due to their lesser mass and are more likely to change with the velocity fluctuation so small diameter droplet simulation will not give exact predictions and mostly it will shows zero collection efficiency.

Also experimental results of droplet collection efficiency are obtained for 3D geometry and simulation is carried out on 2D geometry. Whenever water droplet enters inside vane profile then some droplets will also get stick on wall in 3D geometry and will be drained by gravity from the wall. However in 2D simulations we have considered only hooks as escape boundary for drainage and not for wall. This may show some variation in simulation as compared with experimental result. We can conclude that after simulation result for smaller droplet size result will not vary with experimental value and collection efficiency will depend upon drainage hook geometry. However, the simulation results of 2 dimensional cases show a similar pattern of exponential rise in collection efficiency beyond a particular droplet diameter. The nature of the experimental and simulated values of collection efficiency show by and large a constant difference in all cases, which can be attributed to the use of 2D geometry in simulation instead of 3D geometry and some real life situations.

ACKNOWLEDGMENT

First of all, I would like to express my deepest gratitude to my college guide Prof G. S Barpande and company guide Mr Vitthal Patil for outstanding advice in problems related to this project.I would also like to express my sincere thanks to our head of department mechanical engineering prof. P.b. Joshi and M.E. coordinator prof. Dr. Mrs desai for their valuable guidance and providing all the necessary facilities, which are indispensable in the completion of this project. I am also thankful to the staff members of Mechanical Engineering Department., Finally I express my gratitude towards my Mom and Dad and friends for their valuable support.

REFERENCES

[1] W.Wang, G.A. Davies, CFD studies of separation of mists fromgases using vane-type separators, Trans. I Chem. Eng. A 74 (1996) 232–238.

[2] C. Galletti, E. Brunazzi, L. Tognotti, A numerical model for gas flow and droplet motion in wave-plate mist eliminators with drainage channels, Chem. Eng. Sci. 63 (2008) 5639–5652.

[3] S. Ghetti, Investigation of Entrainment Phenomena in Inertial Separators, University of Pisa, Pisa, Italy, 2003.

[4] P.W. James, B.J. Azzopardi, Y.Wang, J.P. Hughes, Amodel for liquid film flow and separation in a wave-plate mist eliminator, Chem. Eng. Res. Des. 83 (A5) (2005) 469–477.

[5] L.I. Jia, H. Sufi, X. Wang Chin, Numerical study of steam-water separators with wave-type vanes, J. Chem. Eng. 15 (4) (2007) 492–498.

[6] R. Rafee, H. Rahimzadeha, G. Ahmadib, Numerical simulations of airflow and droplet transport in a wave-plate mist eliminator, Chem. Eng. Res. Des. 88 (2010) 1393–1404.

[7] A.I. Josang, M.C. Melaaen, Fluid flow simulations of a vane separator, Model. Identif. Control 23 (1) (2002) 5–26.

[8] R. Rafee, H. Rahimzadeh, Performance evaluation of a curved type vane separator at different plate spacing in the range of 25 to 35 mm using numerical simulation, Iran J. Chem. Eng. 29 (3) (2010) 95–108.

[9] A.I. Josang, Numerical and experimental studies of droplet-gas flow, (Ph.D. Thesis) Department of Technol (HiT-TF), Telemark University Coll, Norway, 2002.

[10] G. Venkatesan, N. Kulasekharan, S. Iniyan, Influence of turbulence models on the performance prediction of flow through curved vane demisters, Desalination 329 (2013) 19–28.

[11] Ansys Fluent 14.0 User's Manual, ANSYS Inc., Canonsburg, PA, 2011.

[12] S.V. Patankar, Numerical Heat Transfer and Fluid Flow, Hemisphere, Washington D.C. 1980.

[13] A. Li, G. Ahmadi, Dispersion and deposition of spherical particles from point sources in a turbulent channel flow, Aerosol Sci. Technol. 16 (1992) 209–226.

[13] *Fatemeh Kavousia, Yaghoub Behjatb, Shahrokh Shahhosseinia,* Optimal design of drainage channel geometry parameters in vane demister liquid–gas separators, chemical engineering research and design 9 1 (2 0 1 3) 1212–1222

[14] G. Venkatesan a, N. Kulasekharan b, S. Iniyan c , Numerical analysis of curved vane demisters in estimating water droplet separation efficiency, Desalination 339 (2014) 40–53

[15] Hans K. Koopmana,∗ , C¸ a˘gatay Köksoyb, Özgür Ertunc¸b, Hermann Lienhartb,Heinz Hedwigb, Antonio Delgadob An analytical model for droplet separation in vane separators andmeasurements of grade efficiency and pressure drop, Nuclear Engineering and Design 276 (2014) 98–106

[16] *R. Rafeea, H. Rahimzadeha,, G. Ahmadib,* Numerical simulations of airflow and droplet transport in a wave-plate mist eliminator , chemical engineering research and design 8 8 (2010) 1393–1404

