Design & Development of Viscous Fluid Mixing Nozzle of Dispensing Machine for Automotive Applications

Mr. Sandip K. Bidgar
PG Scholar, Dept of Mechanical Engineering
AISSMS COE, Pune.

Prof. V Kumar
Dept of Mechanical Engineering,
AISSMS COE, Pune.

Prof. S. V. Chaitanya
Dept of Mechanical Engineering,
AISSMS COE, Pune.

Abstract

Viscous Fluid like Adhesive and Additives has widespread applications in many industries including automotive, aerospace, ship building manufacturing. The goal of CFD analysis of viscous fluid mixing nozzle is to study Grid Patterns and Pathlines of mixing fluid of Dispensing Machine. Dispensing Machine used in a viscous fluid mixing to provide accurate blend ratios in producing the desired end-use product for automotive applications.

The focus of the present paper is to design mixing nozzle for viscous fluid & observe the differenting boundary zones for variable checks like velocity, static pressure, dynamic pressure and density. The main aim of paper is to give accurate mixing of viscous fluid with require pressure of fluid at the outlet of Nozzle for various automotive applications. For proper mixing of viscous fluid eight passes of Nozzle study was done. Computational Fluid Dynamics method has been used to analyze this complex event. Study the Results of CFD Analysis. The study is done with the help of Computational Fluid Dynamics of Mixing Nozzle system using ANSYS FLUENT software.

Keywords: *CFD Analysis, ANSYS FLUNET, Viscous Fluid, Mixing Nozzle, Dispensing Machine, Automotive Applications*

Introduction

Increasingly, high-performance adhesives are being used in the automotive industry to join disparate parts. Adhesive products are used to create a bond between two different or similar materials. Adhesives are made from precise blends of resins, synthetic rubber elastomers, and agents or additives used to enhance certain characteristics, depending on the end use. Adhesives find several applications in an automobile. The applications for adhesives in the automobile industry are as many and varied as the adhesive technologies themselves. Adhesives are used to assemble metal, glass, plastics, rubber, and a variety of other materials during the manufacture of automobiles. Adhesive bonding is increasingly replacing conventional joining techniques such as welding, riveting or bolting. Adhesive bonding renders the vehicles lighter, quieter, more durable and safer. This not only increases safety but also simplifies many production operations, thus enhancing cost efficiency. They are deployed in the manufacture of body components, engines, gear boxes and mounted parts such as wing mirrors, seats and steering wheels where these require pre-treatment, adhesive bonding or coating. Importance of adhesives is make lightweight construction of automobiles. Adhesive bonding reduces steps in the manufacturing processes, which ultimately results in reduced cost.

Another important advantage of adhesives is the relatively high speed with which they can be applied on the production line. As a result, savings can be realised in set-up and processing time and volume of material processed. These potential cost savings are extremely important to car manufacturers and their continuous drive to reduce costs. Consequently, per-vehicle adhesive consumption is likely to remain in upward trend as car designers continue to exploit the cost-saving potential of adhesives. Our environmentally friendly automotive adhesives line provides:

- Improvement of body stiffness and durability
- Functionality over a wide temperature range
- Enhanced crash performance
- Reduction of welding spots
- Greater flexibility in part geometry and material selection

Due to above discussion and benefits, Adhesive are used for various Automotive Applications. The following pie charts show world consumption of adhesives due to its benefits for Automotive Applications.

DESIGN OF MIXING NOZZLE

Design Criteria for Mixing Nozzle:

The static mixer is identified by its diameter and number of mixing elements. For example: MX-1-830 the 8 is the diameter of the housing (1/2") and the 30 is the number of elements (Length) .Most urethanes will mix well with 30 elements although some require 40. Epoxy may mix in as few as 24 elements. Silicon may require a two-stage mixer. To find the right diameter, look at the viscosity of the material being pumped, and the speed. Large fast jobs require a larger mixer. Consider the speed at which the machine will be run. In general, run the machine as slowly as possible to avoid trapping air in the material as it is dispensed. The MX-1-830, or 840, MX-1-648 work well for the medium or fast urethane flow. The optimum design of Helical Mixture is going to be predicted with use of the CFD simulation. CFD will decide the Velocity Profiles inside the Mixture at different Intervals/no of Elements, Velocity profiles in Axial and Radial direction of both the Gel inside the Mixture ,Mass outflow ,Pressure Distribution and concentration of both the gel after mixing. These design criteria will decide the height, Dia, width and no of elements of the Static Mixture.

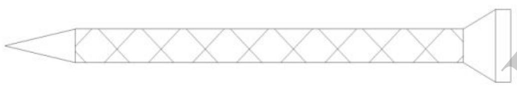


Fig. Design Criteria of Nozzle

Static Mixture (Nozzle Type):-

Mixing is the most fundamental process among all industrial chemical processes, ranging from simple blending to mixing of complex multiphase reaction systems. Improper mixing often results in lower product quality and non-reproducible processing conditions. Consequently, it is then necessary to repeat the procedure of mixing or to use advanced treatment processes, which increases the cost of waste disposal. In many cases static or dynamic mixers are widely used for mixing immiscible liquids. However, static mixers are applied more often than stirrers due to lower operating costs

A liquid-liquid dispersion in static mixers is achieved by passing two immiscible liquids concurrently through the mixers. Detailed knowledge of hydrodynamics in such devices is therefore necessary and it is also vital for the control of the mixing process in the most efficient way.

Characteristic of Static Mixture:-

Static mixers are simple and versatile pieces of equipment often used in continuous mixing, heat transfer, and chemical reactions applications. They have good mixing characteristics, and since they have no moving parts, they can save capital and operating costs. Usually they are designed to disperse and mix two or more fluids in a short length of tube. Improved transport processes occurring in the static mixer such as flow division, radial eddying, flow constriction, and shear reversal eliminate the gradients in concentration, velocity and temperature. Static mixer is a device consisting of a series of flow orientation elements inserted along the axis of pipe. Pressure drives the fluid through the device, providing the energy needed to accomplish mixing. It has the advantage of being easy to install in existing systems. Pressure drop penalties caused by the destructive devices may be large

Laminar Static Mixer Analysis-Technical Challenge

Laminar static mixers work by periodically dividing and recombining fluids until they are well mixed. The performance of these mixers is impossible to analyze without a pure CFD approach due to the limited molecular diffusion between the mixed fluids.

Fig. Various Nozzles

CFD of Static Mixture:-

In recent years an analysis of mixing processes is more frequently conducted by means of CFD method (CFD, Computational Fluid Dynamics). CFD in general allows one to model motion of broadly understood fluids. It is a numerical method in which the conservation laws of mass, momentum and energy, represented in form of transport equations for basic flow quantities (such as velocity, temperature or species concentration) are solved for small control volumes or numerical cells. These control volumes are discretized representation of the continuous fluid. In general the transport equations are usually not sufficient to model complex flows properly due to closure problems (presence of higher order terms in equations or specificity of the physics involved in more complex flows). Therefore for such problems further modeling is required.

Design of Static Mixture using CFD:-

In this work, the flow visualization in a Multiple (20-40)-helical static mixer has been investigated in numerically. The static mixer used for the present study Consists of 20-40 elements with a series of alternating left and right helical elements. Each element has the aspect ratio of 1.5 (i.e., L/D = 1.5) and twisted angle of 180°. The objective of this investigation is mainly to explore the flow pattern inside a helical static mixer due to its novelty and originality. It is clearly observed that the flows at the trailing edge for both numerical lobe-like pattern. Apparently, the numerical solution matches with desired result.

However, fluid flow and mixing performance in theses device has not been rigorously characterized. The scarcity of information about flow and mixing in static mixers is partially available due to their often-complex construction, which makes direct, non-intrusive experimental investigations difficult. The analytical solutions for velocity fields are also impractical due to the complex geometry of static mixers. Nevertheless, the static mixer has increased in popularity within the chemical industry over recent years. Because of a wide range of applications for static mixer, a variety of segment designs is available. Despite widespread usage, the way these mixers work is still not fully understood.

Boundary Condition of Static Mixture:-

The design and optimization of mixers are traditionally performed by trial and error, with much depending on previous experience and wide safety margin. Computational fluid dynamics (CFD) is an increasingly effective alternative to speed up equipment design and gain additional fundamental understanding of mixing process. In the present study, the flow is considered to be three dimensional, laminar, steady and incompressible. The properties of the working fluid (water) are also assumed to be constant. The governing equations contain continuity equation and the momentum equations in three directions. The solution to the governing equations is performed by using finite-volume method of upwind difference scheme. Both continuity equation and momentum equations are solved with solid boundary of no-slip condition. Entrance condition is assigned the entering velocity which is obtained with required Reynolds number. Exit condition is set as outlet of the flow. The model domain of present study is divided into a number of cells as control volumes. In the finite volume method, the governing equations are numerically integrated over each of these computational cells or control volumes. The simulations were performed by using commercial software of ANSYS-FLUENT with version 13.0. The convergence criteria are set as the residual is less than 10-5. Pro-surf (grid generator) is used to generate grid on the surfaces of helical static mixer.

CFD Out Put of Static Mixture:-

For the present study, numerical calculations are performed at specified Reynolds numbers of both fluids. The axial velocity, radial velocity, and vorticity of the fluid are obtained and presented for four different axial locations. Also we predict the cross-sectional distribution of axial velocity, radial velocity, and vorticity, respectively. These properties are presented through the computation domain at four different locations along the axial direction. The four locations are: (i) in the middle of the third mixing element; (ii) at the intersection of the third and fourth mixing elements; (iii) in the middle of the fourth mixing element; and (iv) at the trailing edge of the fourth mixing element

Also we predicts Radial velocity distribution at various locations; (a) middle of third element; (b) at the intersection of the third and fourth mixing elements; (c) in the middle of the fourth mixing element; and (d) at the trailing edge of the fourth mixing element.

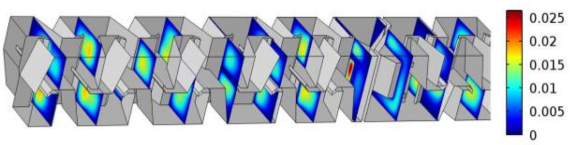


Fig. Velocity profile at different sections along static mixer

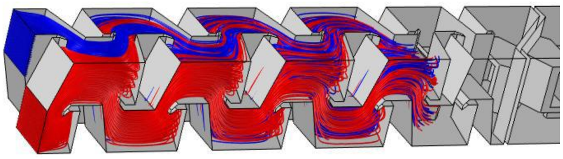


Fig. Streamline plot showing fluid mixing at first 4 mixing elements

Optimized Mixer Designs:

The CFD simulation, particle tracing, and data processing algorithm provide a valuable tool for understanding the flow patterns inside the mixer beyond what is possible by pure experimentation. This is helping us optimize some of their mixer designs.

Viscous Fluids Used for Adhesive Automotive Applications is below

Fluid 1 - Ethyl Cyanoacrylate		
Material properties	Unit	Value
Density	Kg/m^3	1060
Viscosity	Cps	90

Fluid 2 - Polyurethane		
Material properties	Unit	Value
Density	Kg/m^3	1410
Viscosity	Cps	120

Material Used for Nozzle:

For giving above requirements I take standard Nozzle available in market with eight passes such that it satisfied all requirement of end user for Automotive Applications. Material taken for Nozzle used for Mixing Viscous Fluid which is used for various automotive applications is given in following table.

Material: SS304

Material properties	Unit	Value
Young's Modulus	GPa	193
Density	Kg/m^3	8000
Poisson's Ratio	---	0.3
Tensile Strength	MPa	515
Yield Strength	MPa	205

Following Parameters are taken for Nozzle Design. Nozzle Selection is based on Velocity, Viscosity of the fluid. Pouring Pressure of Fluid – 1.5 bar, Inlet Velocity – 35 m/s, Outlet Velocity – 50 m/s, we get Mass flow from Nozzle – 5.86 Kg/s

Design of Nozzle: Following Figure Shows the 3D Drawing of Mixing Nozzle of Viscous Fluid.

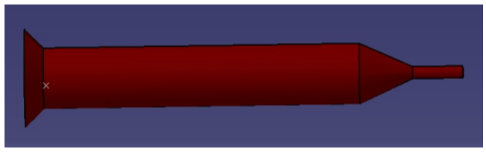


Fig. 3D view of Nozzle

Following Fig. Shows the Section View of 3D Drawing Nozzle which gives clear idea of mixing of viscous fluid. In this design eight passes of fluid is design for proper mixing which gives proper blend ratio of viscous fluid for automotive applications.

Fig. 3D Section View of Nozzle

Following Figure Shows the Details 2D Drawing of Design of Mixing Nozzle which has Eight Passes for proper mixing of Viscous Fluid for various Automotive Applications.

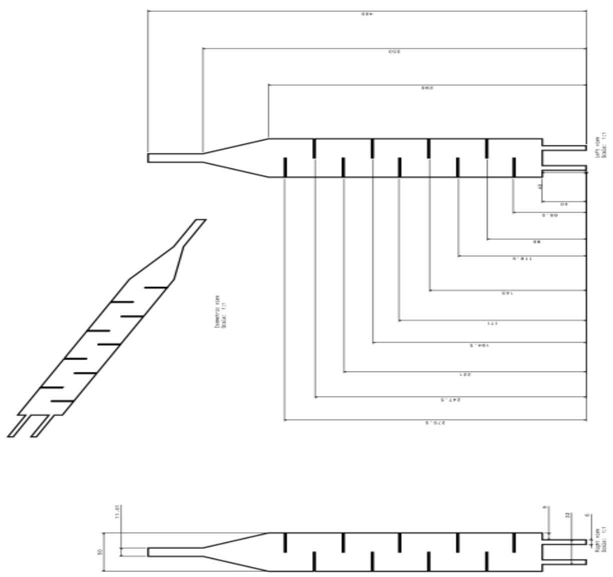


Fig. 2D Details of Nozzle

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 1, January - 2014

COMPUTATIONAL APPROACH

CFD Analysis -

In the computational approach data concerning flow field inside the Nozzle is visualized by simulating the flow condition using ANSYS as the pre-processing software and ANSYS fluent as the solver and postprocessor.

CFD codes have three basic elements which divide the complete analysis of numerical experiment to perform on the specific geometry. The three basic elements are pre-processor, solver, postprocessor. The pre-processing stage of the CFD process involves the following:

Definition of geometry, Meshing, definition of fluid continuum & boundary conditions.

In pre-processing the geometric model of Nozzle is created and saved in the form of .iges file that is input to the ANSYS by means of user friendly software (CATIA).

Fig. Model of Nozzle

In meshing, the region of interest need to be divided into several structure elements using ANSYS as software & grid is generated automatically in 3 dimensional domain with tetrahedral mesh. This is very important stage in CFD as it affects to the accuracy of results Following Figure shows meshed model for Static pressure, Density and Velocity.

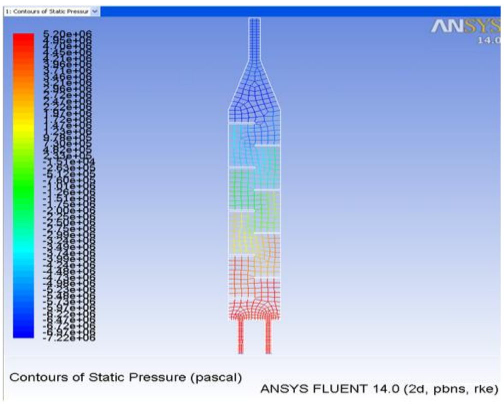


Fig. Contour of Static pressure with mesh

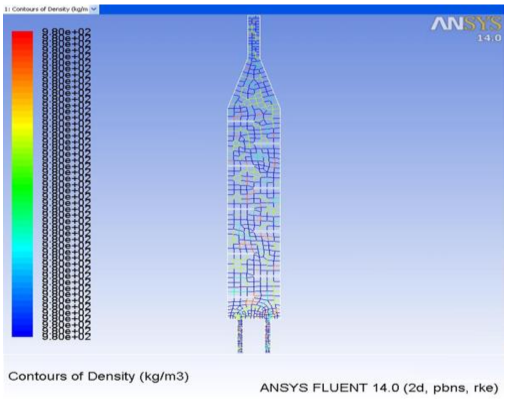


Fig. Contour of Density with mesh

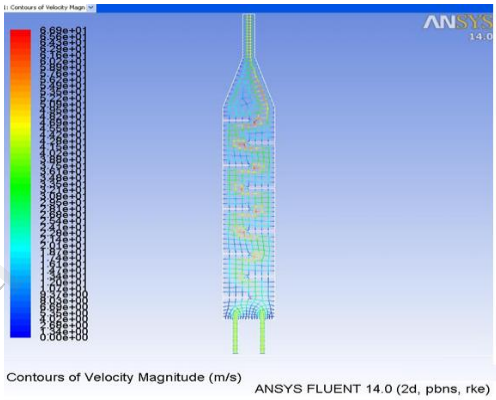
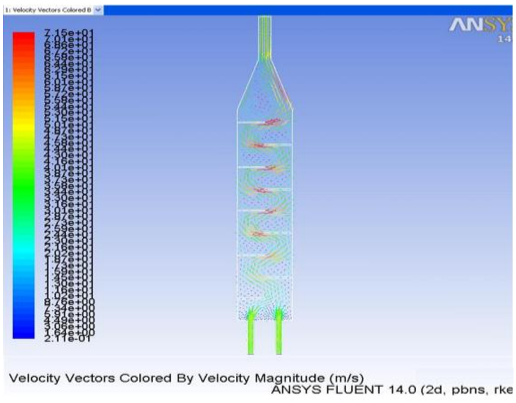


Fig. Contour of Velocity with mesh

Numerical solver: The fluent which is numerical solver is key element of CFD process. The steps involved in this solving process are usually carried out in following sequence. Formal integration of governing equation of fluid flow over all the control volume, conversion of integrated forms of equations by algebraic equations and then calculations of algebraic equations by an iterative method.

Post processing is the last phase of CFD i.e. results and simulation. Contours of velocity by velocity magnitude and static pressure can be obtained from display option. Figure 8.5 & 8.6 shows it respectively for judging viscous fluid output from Mixing Nozzle.



2

Fig. Contours of Velocity Vectors by Velocity Magnitude

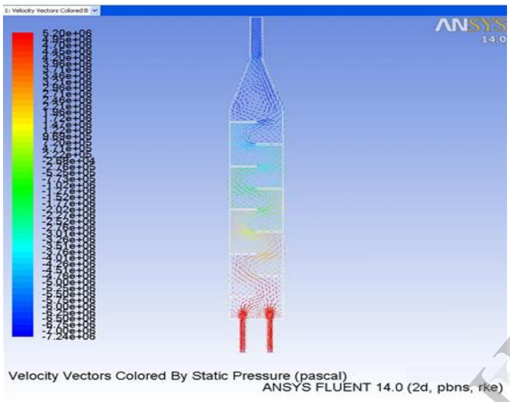


Fig. Contours of Velocity Vectors by Static Pressure

Pathlines are the lines travelled by neutrally buoyant particles in equilibrium with the fluid motion. Pathlines are an excellent tool for visualization of complex three-dimensional flows. In viscous fluid mixing nozzle, Pathlines are used to examine the flow around and in the wake of the Mixing Nozzle.

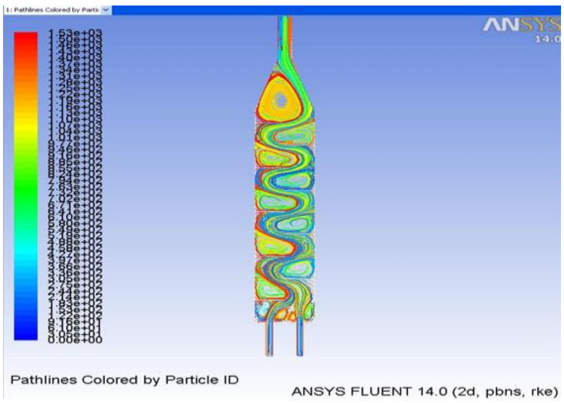


Fig. Pathlines Display

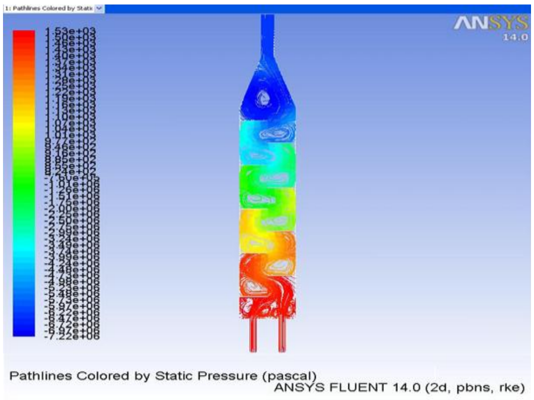


Fig. Pathlines Display

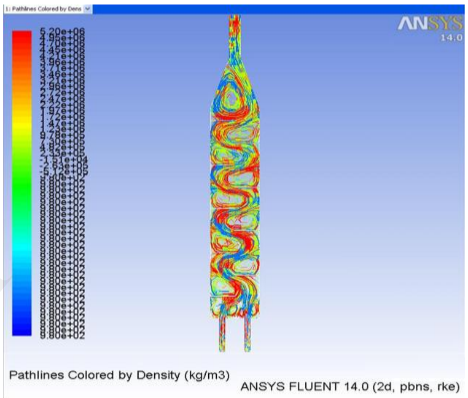


Fig. Pathlines Display

Conclusion

CFD analysis shows the study the computational behaviour of mixing fluid in the nozzle. Pathlines show the flow around and in the wake of the Mixing Nozzle by velocity, pressure and density variations. Design of nozzle gives proper blend ratio in dispensing machine used for various automotive applications.

References

[1] H. Braess Hermann & U. Seiffert, Handbook of Automotive Engineering, SAE International, Warrendale, SA, 2005.
[2] J. Daily & Donald R. Harleman, Fluid Dynamics, Addison-Wesley Publications, 1966.
[3] Eckhard H. Cordes, '50 Adhesives in the Automotive Industry', Mercedes-Benz AG, Bremen, Germany.