RESEARCH ARTICLE OPEN ACCESS

A FLUID FLOW RATE ENHANCEMENT IN A NOZZLE THROUGH CFD APPROCH

K.PAVAN KUMAR [1], J.MURALI MOHAN [2] D.SRIKANTH [3]

1(Department Of Mech, Asst.Professor, Vignan Institute Of Technology And Science Hyderabad, naren2088@gmail.com)

2 (Department Of Mech, Asst. Professor, St Mary's Group Of Institutions, Hyderabad, muralimohan014@gmail.com)

3(Department Of Mech, Asst. Professor, St Mary's Group Of Institutions, Hyderabad, , srikanth.duvvala@gmail.com)

Abstract:

A pressure vessel typically consists of large cylindrical and / or spherical containers with nozzles through which the reactants flow in and out. . While plain cylindrical or spherical containers can be analyzed for internal pressure using thin/ thick cylinder formulae, the ones with nozzles are difficult to analyze. This is in view of complicated stress concentrations that arise at the interface of the nozzle and pressure vessel junction. The calculations have become complicated because of forces that arise at the free end of the nozzle. The forces include those of piping, wind forces, earth quake forces in addition to the internal pressure. In spite of these, strict adherence to safety codes is to be followed. ASME, Section VIII specifies the stress limits to be adhered to. One of the criterions is the stress intensity, which is not possible to compute by simple analytical procedures. FEM can be used for computing the deformation and stress at the nozzle-vessel junction in the structure and also at all other points on the pressure vessel But, precise estimation of stress intensity is not possible with these elements for a structure with nozzles. A method is developed for a precise structured modeling and for estimating the stress intensities at the junction of nozzles and pressure vessels the presence of a ring significantly increases both the turbulence intensity and mean velocity at the exit, and requires a much higher inlet pressure to move the fluid through the nozzle. On the other hand, cutting a groove near the exit or extending the nozzle has little effect on the exit flow characteristics.

Keywords — **Nozzel, FEM, ASME.**

INTRODUCTION

Nozzles are widely used in connection with many different engineering applications, mainly to generate jets and sprays. In most laboratory studies dealing with jets, the emphasis has been on designing nozzles which create a uniform flow with low turbulence intensity at the nozzle exit. On the other hand, in mixing applications, higher turbulence intensity at the exit may be more desirable. The nozzle exit flow serves as the initial condition for the downstream flow. In most instances, flow non-uniformity and turbulence originate within the nozzle, but the nozzle contraction is generally designed to attenuate and minimize these effects To assess the importance of controlling separation of flow inside the nozzle, performed calculations on incompressible flow through nozzles with different inlet curvatures. He concluded that the boundary layer grows less rapidly before a contraction with a smaller inlet curvature due to the milder increase in wall pressure coefficient. Thus, the nozzle can experience a larger change in pressure before the adverse pressure gradient becomes severe enough to cause flow separation, permitting a shorter contraction length. carried out experiments in air on nozzles with different contraction shapes. They found that the mean velocity and the turbulence intensity do not vary across 90% of the exit plane and are independent of the contraction shape, while the velocity and turbulence intensities at the exit plane in the boundary layer region close to the wall are distinct

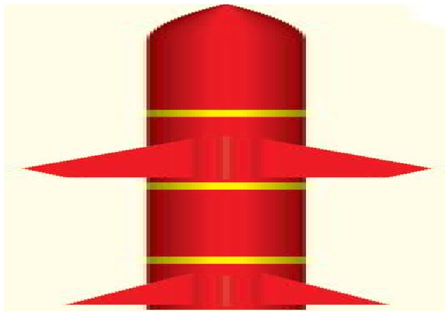

Figure: rocket nozzle

Functions of the contraction shape. Investigated the effect of initial condition on a round free jet. They found that the jet issuing from a contraction nozzle developed faster than that issuing from a straight pipe, and attributed this to the differences in both the near and far field turbulent structures in these flows. Although they did not consider the internal flow within the nozzle, their work clearly illustrates the significance of the initial condition and therefore the importance of characterizing the flow exiting the nozzle.

Governing equation:

A multifluid model based on the Eulerian–Eulerian approach, as used by other similar works was used in this work to describe the fluid flow across the nozzle. This model was assumed to be capable to describe the simple shear flow in the 2D domain. In addition, the 2D domain was sufficient to describe the flow in the nozzle because it is symmetric in the radial direction. Because the mass transfer between the two phases was not considered in the current work, the continuity equation for the system is expressed as

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho u) = 0 \qquad (1)$$

$$\frac{\partial}{\partial t}(\alpha_l \rho_l) + \nabla \cdot (\alpha_l \rho_l u) = 0 \qquad (2)$$

In the current work, the formation of bubble nuclei was assumed to have an insignificant effect on the hydrodynamics in the computational domain. Therefore, the source term that accounts for bubble nuclei was not included in Equations (1) and (2). The calculation for bubble nuclei formation and distribution was performed by using Population Balance Model based on Classical Nucleation Theory

2.0Literature review:

Li, C., & Li, Y. Z. (2011) The entrainment behavior and performance of gas–liquid ejectors computational fluid dynamics (CFD) model and the corresponding algorithm are developed and validation experiment has been carried out over a wide range of operation conditions for ejector with different configurations. Good agreement has been achieved between the predicted values from CFD simulation and the actual data by experimental measurement. The flow patterns that occur within the primary nozzle region are analyzed on the basis of one-dimensional isentropic compressible flow theory.

Payri, R., Ruiz, S., Gimeno, J., &Martí-Aldaraví, P. (2015). Capable of calculating liquid and/or gas problems has been developed, verified and validated. Compressible solvers in Computational Fluid Dynamics use both mass flux and volumetric fluxes through the cell surface to calculate derivative terms. These fluxes depend on density and velocity fields, therefore the stability of the solver is affected by "how" and "where" density and velocity are calculated or updated. In addition to verification and validation, this paper deals with how different flux updates-equations sequences change the computational solution, reaching the conclusion that for mono-phase solvers no extra-updates should be used in order to minimize computational cost, but for multi-phase solvers with high density gradients an extra-update should be implemented to improve the stability of the solver Verification of a new CFD compressible segregated and multi-phase solver with different flux updates-equations sequences.

S. SambhuPrasad , G. Satish , G.Panduranga (2015)the computational fluid dynamics analysis of flow in sudden enlargement and contraction pipes. This project describes an analytical approach to describe the areas where Pipes (used for flow) are mostly susceptible to damage. In this project we discussed to know the pressure values and velocity values at sudden contraction and sudden enlargement of pipes. The software used for this purpose are GAMBIT and FLUENT. The 2D model of the both the pipes are made by GAMBIT and analysis is to be carried out by FLUENTFrom the above analysis, it is observed that the flow is severely disrupted if there are contour changes occurring in the downstream flow in the pipe. Sudden enlargement creates more severe formation of flow eddies than sudden contraction. Also, the losses are more at the point where the enlargement in the pipe begins.

Lahiouel Y., Haddad A., Khezzar(2003),the models are first generated using the data and then are meshed and then various velocity and pressure contours are to be drawn and graphs to analyze the flow through the pipes. Various graphs indicating the variation of velocity, pressure and temperature along the stream length of the pipes are given. Comparisons were made with the sharp corners and smooth corners for the pipeIn the sudden contraction, vena contract's are formed at the point of contraction and this point is the most susceptible point for pipe damage. So, to increase the life of the pipe in cases of sudden contraction the pipes must be designed in view of the above observations making the corners more rounds so as to minimize the losses in the pipes.

Shukla, P., Mandal, R. K., &Ojha,(2001).A mathematical model, based on classical theory of heterogeneous nucleation and volume separation of nucleates among droplets size distribution, is described to predict undercooling of droplets. Newtonian heat flow condition coupled with velocity dependent heat transfer coefficient is used to obtain cooling rate before and after nucleation of droplets. The results indicate that temperature profile of droplets in the spray during resccalescence, segregated and eutectic solidification regimes is dependent on their size and related undercooling. The interface temperature during solidification of undercooled droplets rapidly approaches the liquidus temperature of the alloy with a subsequent decrease in solid–liquid interface velocity. A comparison in cooling rates of atomized powder particles estimated from secondary dendrite arm spacing me

3.0 High temperature materials and their requirement

Recent developments in nuclear power, jet aircrafts, ballistic missiles and rocketry have increased the demand for materials that have good corrosion resistance, strength characteristics and particularly, creep resistance at high temperatures.

NUMERICAL MODELING:

The governing equations for the flow of water in the nozzle are the incompressible Navier-Stokes equations

$$\frac{\partial u_i}{\partial x_i} = 0$$

$$\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j}$$

whereui, p, ρ and ν denote the velocity, pressure, density and kinematic viscosity of water, respectively, and i = 1, 2, 3. In RANS simulations, like those in the current study, averaging of the nonlinear terms in the Navies-Stokes equations results in additional unknown quantities, referred to as the Reynolds stresses.Reynolds stresses, several different turbulence models have been tested and compared using the commercial These turbulence models include the SpalartAllmaras one-equation model, the k–ε standard, realizable and renormalized group (RNG) models, k–ω standard and SST models and the Reynolds Stress Model (RSM) (Wilcox, 1994). Most RANS turbulence models can accurately predict the mean flow quantities and therefore the simpler k - ε standard model is often adequate for industrial applications. However, one cannot assume that the one- or two-equation RANS models are appropriate for the flow in contraction nozzles with high curvature. The computational domain consists of the interior region of the nozzle, as described above. The inner surface of the nozzle is considered to be a no-slip boundary. A uniform velocity of 0.18 m/s with a low level of turbulence is applied at the inlet for all four nozzles. Since the simulation is intended to model flow through a submerged nozzle, with the water entering a tank after exiting the nozzle, a constant pressure outlet condition is applied at the exit. Based on the height of the fluid above the exit of the nozzle, the exit pressure is taken to be about 3 kPa

Validation and verification of numerical model:

Before proceeding to simulate the flow in the four nozzles described above, it is important to validate the numerical model and verify the accuracy of the simulations. Since the cubic equation nozzle of is similar to our baseline nozzle, we use the experimental results for their nozzle, obtained from hot-wire anemometry, to validate out numerical model. Their cubic equation nozzle has an inlet diameter of 254 mm, exit diameter of 76.2 mm, length from tangential point of preconstruction to post-contraction of 152 mm, a straight exit section of 19.5 mm and a total length of 209.6 mm. The Reynolds number in their experiments was 140,000 based on the jet exit velocity (27.4 m/s) and nozzle exit diameter, and the fluid medium was air. The validation has been carried out by comparing the mean and fluctuating quantities obtained using different mesh sizes and topologies and different turbulence models against their data.

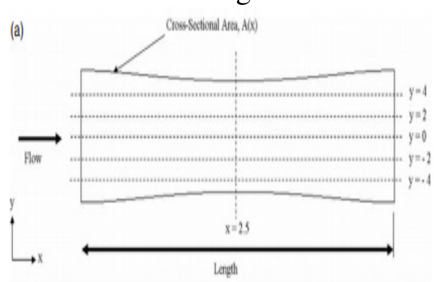

Figure: Modeling view of nozzle

Material properties:

In order to withstand high pressure and temperatures, its strength is acquired by the composition of these materials like Carbon, Nickel, Sulphur, Manganese, Titanium, Silicon and Molybdenum etc. It increases the strength and toughness of the steel. Nickel contributes great strength and hardness with high elastic limit, good ductility and good resistance to corrosion. An alloy containing 25% of Nickel possesses maximum toughness and offers greater resistance to rusting, to rust in Corrosion and burning high temperature. It has zero coefficient of expansion.

MECHANICAL PROPERTIES

Tensile Strength, yield : 750 N/mm^2
Modulus of elasticity : 1.85e5 N/mm^2
Poisson's Ratio : 0.3
Elongation: 18%
Ultimate tensile Strength : 965 N/mm^2

The domain could be computed. The number of molecules is a very important parameter that greatly affects the work of formation of the critical cluster and, consequently, the bubble nucleation rate. The equation for the calculation of the number of molecules required to form a critical cluster was found to depend on the pressure at a specific position. Therefore, the hydrodynamics profile in the nozzle is important to determine the bubble nucleation rate. For comparison, two additional parameters were introduced in the current work. First, a dimensionless parameter, namely, pressure coefficient, Cp, was introduced. This parameter has been widely used in cavitating flow analysis to determine the pressure profile across the nozzle with regards to the inlet flow velocity.

4.0 **results and discussions:**

The CFD simulation coupled with the improved CNT was used to simulate the case of cavitating nozzle flow based on the conditions discussed by Wang and BrennenThe hydrodynamic profiles of the simulated results in the present investigation were compared with the results obtained from Wang and for the case without dissolved air. The comparisons in flow velocity and pressure coefficient along the axis of the nozzle are plotted in Figure 2 and Figure 3, respectively. Based on the figures, the results from the current work exhibit good agreement with the results obtained from Wang and with the highest fluid flow velocity at the throat of the nozzle recorded as 14 m/s, while the lowest

CFD analysis:

Using the approach of coupling CFD and bubble nucleation theory in the 2D computational domain, the distribution of various parameters, including the bubble nucleation rate, pressure, and number of bubble nuclei, can be estimated both in the axial direction and the radial direction. The bubble nucleation rate is strongly dependent on the pressure drop across the nozzle, whereby the hydrodynamics information can be obtained through the CFD approach. In this section, the distribution of pressure and velocity, the number of molecules required to form a critical cluster, and the bubble nucleation rate are presented.

Velocity and pressure:

The contour plots of the velocity distribution for water flowing across the nozzle are shown in Figure 5. Based on Figure 5, the water flows in at a velocity of 10 m/s and the speed increases at the throat of the nozzle to nearly 14 m/s due to the constriction. Subsequently, the speed of the water flow gradually reduces again to approximately 10 m/s. The increase and decrease of velocity of the water was predicted based on the laws of mass and momentum conservation, with the assumption of an incompressible fluid flow. (a)shows the contour plot of the pressure distribution in the nozzle. Based on the law of momentum conservation because there was no external force applied to the fluid flow and because of the difference in height, theincrease of velocity in fluid flow must be compensated by a pressure drop. As a consequence of the increase of the velocity of the water flowed across the nozzle, the pressure of the water at the throat of the nozzle has dropped to nearly –46 kPa in the current work, that is, as if the nozzle was being stretched

Boundary Conditions:

Velocity inlet was taken for the nozzle inlet and the value of velocity inlet was taken as 2m/sec. Initial gauge pressure was taken as 101325 Pascal. Temperature was taken as 500K.The solution is set as listed below. The under relaxation factor was set as givenPressure-0.3 Density-1 Body forces-1 Momentum-0.7

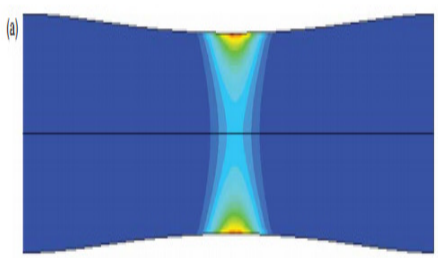

Figure:contour plot across the nozzle

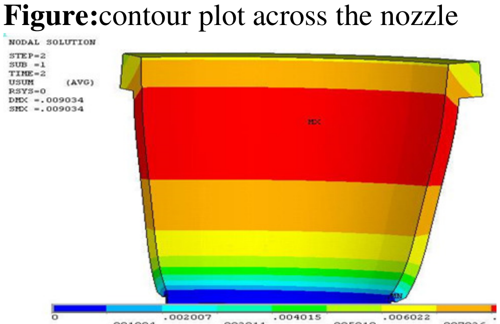

Figure:displacement of cone for load

From the Resultant displacement contour maximum displacement is 0.0903 mm. Maximum displacements in the Nozzle cone is due to, component is assembled at the free end of Nozzle. Minimum displacement is 0.00100 mm in the nozzle cone.

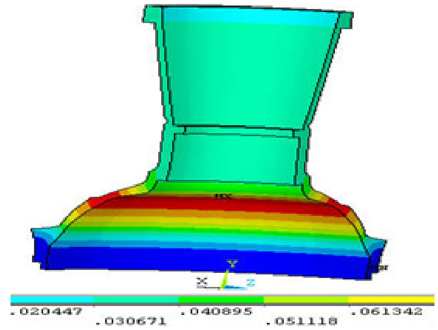

Figure:displacement of nozzle assembly applying thrust for load

Table :Mass Flow Rate

mass flow rate	kg/s
Default-Interior	79557.105
Inlet	79856.001
Outlet	79854.269
Wall	0

Conclusions:

Although most turbulence closure models reasonably predict the mean quantities for the flow through a nozzle, the Reynolds Stress Model shows significant improvement over the standard one-equation and two-equation models for the prediction of the turbulent flow parameters. Mesh topology also plays a major role in determining the accuracy of the results when compared to experimental data. From the above analysis, it is observed that the flow is severely disrupted if there are contour changes occurring in the downstream flow in the pipe. Sudden enlargement creates more severe formation of flow eddies than sudden contraction. Also, the losses are more at the point where the enlargement in the pipe beginsIn the sudden contraction, vena contracts are formed at the point of contraction and this point is the most susceptible point for pipe damage. So, to increase the life of the pipe in cases of sudden contraction the pipes must be designed in view of the above observations making the corners more rounds so as to minimize the losses in the pipes.

References:

[1] Li, C., & Li, Y. Z. (2011) Investigation of entrainment behavior and characteristics of gas–

liquid ejectors based on CFD simulation. Chemical Engineering Science, 66(3), 405–416.

[2] Payri, R., Ruiz, S., Gimeno, J., &Martí-Aldaraví, P. (2015). Verification of a new CFD compressible segregated and multi-phase solver with different flux updates-equations sequences. Applied Mathematical Modelling, 39(2), 851– 861.

[3] S. SambhuPrasad , G. Satish , G.Panduranga (2015) Comparison of Flow Analysis through Sudden Contraction and Enlargement of Pipes by Providing Smooth Corners International Journal of Engineering Trends and Technology (IJETT) – Volume 25.

[4] Lahiouel Y., Haddad A., Khezzar (2003), Development of method of routing fluid distribution network, J. Mechanica, Vol. 43, pp. 27-34.

[5] Shukla, P., Mandal, R. K., &Ojha(2001)Non-equilibrium solidification of undercooled droplets during atomization process. Bulletin of Materials Science, 24(5), 547–554.

[6]Lahiouel Y., Haddad A., Chaoui K., ―Evaluation of head losses in fluid Transportation networks Sciences & Technologies B – N°23, juin (2005), pp. 89-94.