OPTIMIZING THE VELOCITY OF RING SHAPE PARAMETER FOR DESIGNING THE NOZZLES USING CFD

Obai Younis1,2 *, Reem Ahmed3, Ali Mohammed Hamdan4, Dania Ahmed2
1Prince Sattam Bin Abdulaziz University, College of Engineering at Wadi Addwaser, Department of Mechanical Engineering, Wadi Addwaser, Saudi Arabia
2University of Khartoum, Faculty of Engineering, Department of Mechanical Engineering, Khartoum, Sudan
3Elgerafsharg Technical College, Department of Mechanical Engineering, Sudan
4University of Bahri, Department of Mechanical Engineering, Alkadroo, Sudan

This study aims to optimize the velocity of ring shape parameter for designing the nozzles using computational fluid dynamics (CFD) and investigated the flow in nozzles using ANSYS, Inc. simulation software. The model geometries were defined using ANSYS FLUENT-Design Modeler platform. All nozzles were designed on unstructured triangular elements comprising of 1200000 mesh nodes. The differential governing equations were applied in ANSYS FLUENT based on a finite volume method. The distance and dimensions of ring location significantly influence the velocity of water during flow where the maximum velocity at double rings reduces the surface area at distance of 7mm and 15mm and 2x2 mm dimensions. Considering 8, 10, and 12 bar liner proportions, there was an increase in the velocity at maximum points in ring shapes.

Key words: CFD, nozzles, optimization, ring shape parameter, velocity.

INTRODUCTION

The fluid dynamics problems are typically complicated for solving as the system of equations is strongly non-linear system in order to govern the phenomenon. It becomes complicated for finding accurate solutions. On the contrary, computational fluid dynamics (CFD) technology was successful in order to calculate the fluid dynamics because of the close proximity between numerical simulation, experimentation, and theory in fluid dynamics. It has been observed from the theory proposed by Back, Massier&Gier [1] that experience is mandatory for testing the hypotheses, whereas this theory becomes important for explaining the findings. Therefore, it is essential for validating the experimental results as numerical simulation is independent of experience.

The effect of nozzle geometry and shape on flow is interacted with numerous nozzle simulations with different types of nozzles and fluids. Fluids in nozzles were examined in previous studies. For instance, the performance of fluid dynamic was studied by Brusiani, Falfari&Pelloni [2] for three different injector hole shapes diesel nozzle cylindrical, k hole, and KS hole by investigating the nozzle layouts through 3D-CFD fully transient multiphase approach. The flow field was studied numerically by Tamaki et al [3] for radial turbine using CFD and variable area nozzle. The study has revealed, by examining two throat areas, that the influence was very weak with the largest opening on leakage flow on the flow field nozzle downstream. Similarly, the impact of entertainment near the nozzle inflow was examined by Babu and Mahesh [4] on spatially evolving turbulent jets and round laminar jets through direct numerical simulation (DNS). The study has revealed that the significance of nozzle inflow facilitates in setting the flow of turbulent jet simulation.

The effect of nozzles geometry was numerically examined by Matsuo et al [5] with the interaction between flow characteristics and nozzle geometry in spiral nozzle via unified platform for Spalart-Allmaras as turbulence model and aerospace computational simulation. Furthermore, Theerayut and Nuntadusit [6] have examined the flow attributes of jet from expansion pipe nozzle through the numerical simulation model and 3-D numerical simulation with standard k-ε turbulent model. The study has revealed that a reverse flow of ambient fluid was generated through these characteristics into the pipe nozzle chamber with the collar. Alam et al [7] have examined the impact of flow parameters at the exit of nozzles for focusing on the impact of nozzle flow parameters. The study has found that the nozzle geometry had an apparent impact on discharge coefficient via Reynolds-averaged Navier-Stokes (RANS) equations.

Belega and Nguyen [8] have examined the nozzle flow by using the convergent-divagated model and identified the fluid behaviour using CFD. Similarly, the flow in convergent-divagated nozzle was examined by Rao et al [9] with different Mach number and nozzle ratio using CFD. In addition, CFD analysis of flow was used by Satyanarayana et al [10] for different cross-sectional shapes in convergent-divagated nozzles in order to examine the appropriate nozzles for giving high-exit velocity among different cross-sections. The study has revealed that the shape of nozzle is dependent on the fluid characteristics for affecting the flow throughout the nozzle and flow expansion level. Moreover, the compressible flow in convergent-divagated nozzle was numerically simulated by Mohamed et al [11] for studying the pressure effect on the flow attributes. The turbulence was modelled by us-
ing Reynolds averaged Navier-Stocks (RANS) equations and two transport equations.

There is a paucity of literature discussing the numerical simulation of flow in nozzles. For instance, the effect of diesel nozzle geometry on the introduction and development of cavitation was numerically examined by Payri et al. [12] and Macian et al. [13]. Du, Liu and Tang [14] have conducted CFD for studying the nozzle internal flow and diesel fuel spray, while Masuda et al. [15] included the mixture formation in the chamber through CFD simulations. A CFD cavitation model was developed by Giannadakis, Gavaises and Arcoumanis [16] for diesel injector nozzles on the basis of Eulerian-Lagrangian approach. Giannadakis and colleagues have explained that their model can express several cavitation structures in internal nozzle flows and revealed that these structures are relied on flow conditions and nozzle design. Shan, Zhang and Huang [17], in a recent numerical and experimental study, have compared the aerodynamic performance of several micro lobe nozzle ejectors with different geometries. Shan, Zhang and Huang [17] have used the k-ε standard turbulence model for conducting RANS simulations.

The majority of nozzle simulations are concerned with the impact of nozzle shape on the association of the spray or jet with the fluid downstream of the nozzle. For instance, direct numerical simulation was undertaken by Boersma, Brethouwer and Nieuwstadt [18] at a low Reynolds number of 2400 using a spatially developing free round. Boersma, Brethouwer and Nieuwstadt [18] have compared their outcomes with the simulations conducted by Hussein, Capp and George [19] and Panchapakesan and Lumley [20] and revealed better accord with the numerical simulations. The impact of inflow conditions was examined by Babu and Mahesh [4] on the inclusion of the ambient fluid into the jet exiting the nozzle. Various scholars have assessed the accuracy of the turbulence model in order to expect the flow field and the nozzle performance by conducting the numerical and experimental studies. These studies have conducted two-dimensional axisymmetric compressible flow analysis using a CD nozzle with the assistance of ANSYS FLUENT via K-ε turbulence model. In this regard, Najar, Dandotiya and Najar [21] have conducted a comparative assessment between the models on the velocity, temperature contours, vectors, and pressure for developing the efficient design conditions for CD nozzles.

There seems to be a lack of basic understanding about the changes made by the flow field with respect to the changes made in the nozzle exit region, although researchers have highlighted the significance of the initial conditions on the downstream flow at the nozzle exit plane. Thereby, it is of significant interest for developing a CFD approach for examining the turbulence and mean features of fluid flow using a nozzle, and for studying the impact of nozzle shape on the exit conditions in the post-contraction region.

CFD is a multipurpose technique of simulation and modelling of flow fields, which offers accurate findings about the flow features of an object. The complexity in the computational studies of the flow field is imposed through the solution of RANS being transient in nature, and the integration of an adequate turbulence model for closure of RANS equations. Discrepancies between the experimental measurements and numerical simulations are induced by the compressible flow regions in nozzles being dominated by complex secondary flows and intense pressure gradients.

It has been observed that different methods of analysis were covered in previous studies related to the flow of nozzles. However, numerical method has described effectively the results as compared to the experimental method, which require complicated nozzle geometries. Current modifications in building sizes and increase have raised concerns for additional water with high velocity, specifically in case of fire, for mitigating the associated obstacles. In this regard, this study has optimized the velocity of ring shape parameter in order to design the nozzles using CFD.

METHODS

The study of fluids in motion is dedicated through CFD and how processes are influenced by the fluid flow behaviour including chemical reactions and heat transfer in combusting flows. In addition, fundamental mathematical equations demonstrate the physical attributes of the fluid motion typically in the form of partial differentiation, which direct a process of interest, and are usually termed as governing equations in CFD. In this study, CFD technique has been used for examining the velocity of ring shape parameter for designing the nozzles.

Generally, the Euler-Euler and Euler-Lagrange approaches were often utilized for numerically simulating the multiphase flows. The continuous phase, in the Euler-Lagrange is modelled by explaining the time aggregated Navier-Stokes equations, whereas the disseminated phase is explained by tracking a series of droplets, bubbles, or particles via the computed continuous flow field. This approach made a fundamental postulation that the disseminated phase possesses a low-volume fraction, which would recommend that the disseminated phase attributes are not closer and must be treated as independent [22, 23].

On the contrary, different phases, in the Euler-Euler approach, are treated mathematically to explain ranges. However, this approach indicates that the phases separate or mix, and possesses a high-volume fraction [23, 24]. The high-volume fraction recommends that the disseminated phase attributes are closer to be considered as independent. Thereby, the engagement between the influence of the secondary phase and multiphase flow will be larger to measure for. The Volume of Fluid, Eulerian-Eulerian, and Mixture models are the three different Euler-Euler multiphase flow models.
This study has used ANSYS, Inc. simulation software for investigating the flow in nozzles. The nozzles were defined using software: SolidWorks in order to introduce ring shapes. The dimensions proposed by Zhang et al., [25] have been used to describe the nozzle base model in this study which is shown in figure (1). This nozzle is 20 mm in diameter at the outlet, 141.96 mm in length, and 50 mm in diameter at the inlet. In addition, this study has presented all nozzles with equal length and inlet and outlet diameters for the nozzles.

The shapes of the physical boundaries of the fluid are specified through the nozzle geometries definition. ANSYS FLUENT-Design Modeler platform was used to define the model geometries. A 2D analysis type was selected for defining the nozzle dimensions based on the model. The shapes and computing areas of nozzles were drawn through ANSYS CFD software.

The total mesh number of nodes (1200000) was adopted for unstructured triangular elements on surfaces for all nozzles as shown in figure (2). Coarse grid solutions might be affected by the mesh topology and element type topology, and in turns achieved a grid-independent solution by affecting the mesh resolution. In addition, the mesh sizes increment was progressively reduced by the number of extension nodes using an exponential growth function where the aspects were positioned away from the outlet and inlet regions. The level of grid sizes from coarse to fine were ascended by the number of elements directly proportional to the numbers of nodes. The study has selected medium mesh level with high smoothing in order to represent the mesh density. It is observed that the turbulence is captured through the fine mesh rather than the interior domain. The number of grids points and computational time was reduced by this possession (Table 1).

![Figure 1: Geometry of the Nozzle](image1)

Figure 1: Geometry of the Nozzle

![Figure 2: mesh generation](image2)

Figure 2: mesh generation

![Figure 3: Flow rate for different mesh number](image3)

Figure 3: Flow rate for different mesh number

Table 1: Mesh type and number of nodes

<table>
<thead>
<tr>
<th>Mesh</th>
<th>coarse</th>
<th>medium</th>
<th>fine</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of nodes</td>
<td>800000</td>
<td>1200000</td>
<td>1400000</td>
</tr>
</tbody>
</table>

Validation is the preliminary mean for evaluating the accuracy and reliability in computational simulations. A well-documented benchmark experimental or numerical data has to be used to validate the process that assess modelling uncertainty. In order to validate the numerical approach utilized in this study, the obtained CFD results were compared with the numerical data of Zhang et al. [25]. Figure 4 presents the comparison between current study and Zhang et al. [25] for axial velocity component. It is clear that a very good agreement between the results of current study and of Zhang et al. [25] is obtained.

![Figure 4: Axial velocity component (current study Vs. Zhang et al. [25])](image4)

Figure 4: Axial velocity component (current study Vs. Zhang et al. [25])
Boundary Conditions

The inlet of nozzles was shown with a pressure of 8 bar, 10 bar, and 12 bar, respectively, and the outlet pressure of 1.10235 bar. No slip condition describes the boundary conditions at the wall including grooves, rings, and nozzle. This refers to the relative velocity between the fluid and wall, which was adjusted at zero. For all models, flow analysis was conducted in steady state mode.

Solver

ANSYS FLUENT 15.0 was selected for the simulations of the CFD technique. With the consideration given to viscous, monophasic, and incompressible fluid and isothermal process fluid motion, CFD simulations were simulated for turbulent flows in steady state. During the numerical computing, a finite volume method was applied to solve the differential governing equations in ANSYS FLUENT. The upwind scheme of second order was performed for the spatial discretization for all conservation equations and for the fluids. Similarly, the pressure and velocity were coupled in this phase using SIMPLE scheme. The second-order accuracy was selected for the overall spatial discretization. A liquid-water is selected as material for the overall spatial discretization.

Following are the parameters selected for the solution controls (Table 2):
- Turbulence Dissipation rate = 0.8
- Turbulence Kinetic Energy = 0.8
- A default under relaxation factor

<table>
<thead>
<tr>
<th>Procedure</th>
<th>Details</th>
</tr>
</thead>
<tbody>
<tr>
<td>Problem set up</td>
<td>Type: pressure-based Velocity; absolute Time: steady 2D space: planar</td>
</tr>
<tr>
<td>General- solver</td>
<td></td>
</tr>
<tr>
<td>Models</td>
<td>Viscous: turbulent SST k-ω</td>
</tr>
<tr>
<td>Materials</td>
<td>liquid, water</td>
</tr>
<tr>
<td>Boundary condition</td>
<td>Inlet: pressure inlet At 8, 10 &amp;12 bar Outlet: pressure outlet</td>
</tr>
<tr>
<td>Reference value</td>
<td>Compute from: inlet Reference zone: solid surface body</td>
</tr>
<tr>
<td>initialization</td>
<td>Hybrid initialization</td>
</tr>
</tbody>
</table>

Through these properties, residual monitoring was ensured whereas convergence criteria were prepared. The iterations continue after adjusting the number of iterations to capture the convergence. The configuration and dimensions of all studied cases are summarized in Table 3.

<table>
<thead>
<tr>
<th>Nozzle Model type (Rings)</th>
<th>Geometries parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td>Model a</td>
<td>One ring with dimensions of 1×1 mm and distance from nozzle exit 10 mm.</td>
</tr>
<tr>
<td>Model b</td>
<td>One ring with dimensions of 1×1 mm and distance from nozzle exit 15 mm.</td>
</tr>
<tr>
<td>Model c</td>
<td>One ring with dimensions of 2×2 mm and distance from nozzle exit 10 mm.</td>
</tr>
<tr>
<td>Model d</td>
<td>One ring with dimensions of 2×2mm and distance from nozzle exit 15 mm.</td>
</tr>
<tr>
<td>Model e</td>
<td>Double rings with dimensions of 1×1 mm, 7mm distance for first ring and 15mm for second ring from nozzle exit.</td>
</tr>
<tr>
<td>Model f</td>
<td>Double rings with dimensions of 2×2 mm, 7mm distance for first ring and 15 mm for second ring from nozzle exit.</td>
</tr>
</tbody>
</table>

RESULTS

The velocity of water during flow was significantly affected by the distance and dimensions of ring location, specifically where the maximum velocity was given by double rings with 2x2 mm dimensions and distance of 7mm and 15mm from nozzle exit in order to reduce the surface area. The optimization was found to be positive where the velocity increased with the increase in pressure. It was also established that the mean velocity was substantially increased with the presence of ring at the exit, and needs a much higher inlet pressure for moving the fluid using the nozzle. In addition, the narrowest outlet section in all nozzle types is reported through the maximum value of velocity at the outlet section.

Similarly, the distance of 7mm and 15mm for the first ring and second ring with 2x2 mm dimensions were selected for changing the ring to double rings. The velocity was increased at maximum point considering at all pressure 8, 10, and 12 bar liner proportion. Due to the number of rings and its associated dimensions, the surface area might be affected because of this change and 2x2 mm dimension.

In addition, Table 4 presents the changes in number of rings to double rings with 7 mm distance for first ring and 15 mm distance for second ring along with 1x1 mm dimensions. It has been observed that variations exist between the increase in velocity at 8 bar, 10 bar, and 12 bar liner proportion, which reduced the surface area.
Table 4: Results analysis for rings

<table>
<thead>
<tr>
<th>Models type (Ring)</th>
<th>Velocity (m/s)</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>At 8 (bar)</td>
<td>At 10 (bar)</td>
<td>At 12 (bar)</td>
</tr>
<tr>
<td>Model a</td>
<td>41.8</td>
<td>46.7</td>
<td>51.1</td>
</tr>
<tr>
<td>Model b</td>
<td>47</td>
<td>52.6</td>
<td>57.7</td>
</tr>
<tr>
<td>Model c</td>
<td>41.1</td>
<td>46.2</td>
<td>50.9</td>
</tr>
<tr>
<td>Model d</td>
<td>42.3</td>
<td>48.5</td>
<td>51.8</td>
</tr>
<tr>
<td>Model e</td>
<td>45.7</td>
<td>51.1</td>
<td>56</td>
</tr>
<tr>
<td>Model f</td>
<td>49.1</td>
<td>55.1</td>
<td>60.6</td>
</tr>
</tbody>
</table>

The effect of ring dimensions 1×1 (mm) located 10 mm at 8 bar, 10 bar, and 12 bar, in velocity of water during flow in the nozzle is shown in Figure 5. The model reveals that the relationship of pressure and effect of ring dimension 1×1 mm and distance is proportion.

Figure 5: Velocity contours of first ring model with dimensions of 1×1 mm and distance from nozzle exit at 10 mm

Figure 6 shows the effect of ring dimensions 2×2 (mm) and location 10 mm at 8 bar, 10 bar and 12 bar, in velocity of water during flow in the nozzle. The model revealed that the relationship of pressure and effect of ring dimension 1×1 mm and distance 15 mm is proportion relationship, and rate of velocity is more than the model presented in Figure 5.
Figure 6: Velocity contours of first second ring model with dimensions of 1×1 mm and distance from nozzle exit at 15 mm

Figure 7 reflects the effect of the ring with 2×2 mm dimensions and distance of ring location is 10 mm far away from exit of the nozzle. It also exhibits its effect in velocity increased at 8 bars, and at 10 bars, while, at 12 bar the velocity increased more than 8 bar and 10 bars. This variation shows that when ring thickness increased the velocity decreased because of water separation caused by distance reduction.
Figure 7: Velocity contours of first third ring model with dimensions of 2×2 mm and distance from nozzle exit at 10 mm

Figure 8 shows the effect of ring dimensions 2×2 (mm) and location 15 mm at 8 bar, 10 bar and 12 bars, in velocity of water during flow in the nozzle. This model reveals that the relationship of pressure and effect of ring dimension 2×2 mm and distance 15 mm is proportion. This may be due to difference in ring dimensions and increase in ring thickness that decreases velocity because of water separation caused by distance reduction.

Figure 8: Velocity contours of fourth ring model with dimensions of 2×2 mm and distance from nozzle exit at 15 mm

Figure 9 illustrates the effect of double rings dimensions 1×1 (mm) and location 7 mm for the first and 15 mm for second ring from the exit at 8 bar, 10 bar and 12 bars, in velocity of water during flow in the nozzle. The results demonstrate that its effect in velocity increased at 8 bar, 10 bar and 12 bar velocity because they at 10 mm distances, unlike Figure 2.
At 8 bars

At 10 bars

At 12 bars

Figure 9. Velocity contours of fifth ring model with dimensions of 1×1 mm, 7mm distance for first ring and 15mm for second ring from nozzle exit

Figure 10 shows the effect of double rings dimensions 2×2 (mm) and location 7 mm for the first and 15mm for second ring from the exit at 8 bar, 10 bar and 12 bars, in velocity of water during flow in the nozzle. The velocity increased at 8 bar and also increased in 10 bar and 12 bars in liner proportion due to number of rings and its dimension that reduced the surface area.
The results have clearly shown that there is significant impact of dimensions and distance of ring location on the velocity of water during flow, especially where double rings with dimensions with 2×2 mm and distance 7mm and 15mm from nozzle exit. This condition offers maximum velocity as the result of decrease in surface area. The rationale for this mechanism can be explained by the presence of large separation that reattaches near the exit of the ringed nozzle. The exit flow increases, along with re-entrant flow near walls due to elongated recirculation zone as the baseline nozzle is stimulated with an added ring.

Therefore, it is stated that velocity increases as the surface area of nozzle decreases. The result stating that there is significant positive effect of increase in pressure on velocity is in agreement with Yu et al. [26]. Moreover, the analysis also showed that presence of a ring significantly increases mean velocity at the exit and increased inlet pressure is required to move the fluid through the nozzle. These results are consistent with one of the previous studies conducted by Bilir et al. [27], who showed that narrow outlet section in all nozzle types helps velocity to reach at its maximum value.

CONCLUSION

The two-dimensional steady flows were simulated for investigating the effect of velocities in the nozzles undertaking different pressures. The study has adjusted twelve monitoring sections in the nozzles horizontally. The core objective of this study was to optimize the ring shape velocities on the water flow. The study has applied the water passing through the nozzle shapes for alleviating the velocities. A homogenous flow assumption and steady simulation was used for simulating the entire flow passage of models through shear stress transport equations along with standard turbulent model. The nozzle was compared with the simulation results regardless of any modification. Better expectations were reflected through these results regarding the use of rings in order to enhance operation of nozzles in the same dimension for increasing water velocity. It was observed that no changes were made in different parameters with the modifications made in the nozzle for grooves. Therefore, considering the results, future studies can utilize these models for increasing the velocities and improving the surface area covered by water. Future studies should consider adopting short distance for decreasing the velocity and separating the water. Additional rings should be applied for gaining more insights on effects of velocities.

ACKNOWLEDGEMENT

This publication was supported by the Deanship of Scientific Research at Prince Sattam bin Abdulaziz University, Alkh., Saudi Arabia.

REFERENCES


Paper accepted: 02.03.2021.
This is an open access article distributed under the CC BY 4.0 terms and conditions.