Design and Fabrication of Water Pressure Nozzle by Using CFD Flow Analysis
*P Praveen Kumar1, N.Nikhil1, D.Srinivas1, P Srikanth Reddy1, B Krishna Teja1
1Department of Mechanical Engineering, Sreyas Institute of Engineering and Technology, Hyderabad, India.
Received- 5 Feb 2018, Revised- 16 March 2018, Accepted- 24 March 2018, Published- 30 March 2018

ABSTRACT
In this article the design of nozzle according to specifications was done. Further nozzle has been analyzed in CFD for determination of different Inlet/outlet properties. Based on the results of the CFD THE actual nozzle were fabricated by using lathe operation such as Turning, facing, Drilling, Tapering, Knurling and Threading and its is tested for determining the velocity and pressure. After the fabrication of the nozzle the flow analysis was carried out by determining the inlet & outlet parameters measured by the theoretical and practical analysis are shown to be approximately same.

Keywords: Nozzle, ANSYS, CFD Fluent.

1. INTRODUCTION

Nozzle as shown in Figure 1 is a device which is used for to increasing the velocity and decreasing the pressure. Nozzle is often a pipe or tube of varying the cross sectional areas, and it is used for changing the direction or modified the flow of fluid. Now a days, there are number of development in Aerospace Engineering, defence, civil and mechanical prospects. Extensive research has been carried out in these fields on flow analysis [1].

This work tries to invent methods of developing the rocket nozzles and high pressure nozzles used in rocket engines for provide high thrust. The test is virtualized at different Mach numbers, like sonic, subsonic, supersonic respectively, where the flow conditions are derived [3]. The virtualization is one of the major developments in the field of research, which is revolution. The computational techniques are used widely for getting better results, close to experimental techniques[2]. The flow through a converging and diverging nozzle is one of the benchmark Problems used for modelling the compressible flow through computational fluid dynamics. Occurrence of shock in the flow field displays one of the most prominent effects of compressibility over fluid flow. Accurate shock predication is a challenge to the CFD fraternity. The pressure grading can be resolved by using special numerical shows. And also by application of fine grid or mesh of the component [4,5].

This study is very much use full in the application of CFD in fabrication of variant types of Nozzle and to predict the flow of fluid.

2. WORKING PRINCIPAL OF NOZZLE

The fluid enters into the nozzle at low velocity and with high pressure. The convergent are and throat area of the nozzle the flow velocity increases and decreases the pressure at the outlet. This phenomena can be attributed by the application of Bernoulli’s equation at various section of the nozzle Equation(1) gives the Bernoulli’s carry at inlet and outlet.

\[
\frac{p1}{pg} + \frac{v1^2}{2g} + z1 = \frac{p2}{pg} + \frac{v2^2}{2g} + z2
\]  

(1)
Where, \( P_1 \)-Initial pressure at inlet  
\( P_2 \)-Final pressure at outlet  
\( \rho \)-Density  
\( g \)-Gravity  
\( V_1 \)- Initial Velocity at inlet  
\( V_2 \)- Final Velocity at outlet  
\( Z \)-Datum head

### 2.1. General Form of Navier – Stokes Equation

The Equation (2) gives the Navier-Stokes equation used for the computing the flow analysis in CFD.

\[
\frac{\partial (\rho \Phi)}{\partial t} + \frac{\partial}{\partial x_i} \left( \rho U_i \Phi - \Gamma_\Phi \frac{\partial \Phi}{\partial x_i} \right) = q_\Phi \quad (2)
\]

It is used to solve or analyze the viscous flow problem using the governing equation. This equation has been utilized for solving the flow rate at different section of the Nozzle. Table 1 Shows various properties of different fluids.

#### Table 1: Shows various properties of different fluids.

<table>
<thead>
<tr>
<th>Substance</th>
<th>Air</th>
<th>Water</th>
<th>Honey</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density (kg/m³)</td>
<td>1.275</td>
<td>1000</td>
<td>1446</td>
</tr>
<tr>
<td>Viscosity</td>
<td>1.82e-4</td>
<td>1.002e-2</td>
<td>190</td>
</tr>
</tbody>
</table>

### 3. METHODOLOGY

#### 3.1. Specifications of Nozzle

The Table 2 and Figure 2 shows the dimension and the specifications used for the design of the nozzle in the CFD platform.

#### Table 2: Geometry dimensions

<table>
<thead>
<tr>
<th>SI No</th>
<th>Name Of The Part</th>
<th>Dimensions</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>Inlet diameter</td>
<td>9.5</td>
<td>mm</td>
</tr>
<tr>
<td>2.</td>
<td>Throat diameter</td>
<td>2.4</td>
<td>mm</td>
</tr>
<tr>
<td>3.</td>
<td>Outlet diameter</td>
<td>2.4</td>
<td>mm</td>
</tr>
<tr>
<td>4.</td>
<td>Length of the throat</td>
<td>10.9</td>
<td>mm</td>
</tr>
<tr>
<td>5.</td>
<td>Length of the inlet</td>
<td>11.4</td>
<td>mm</td>
</tr>
<tr>
<td>6.</td>
<td>Length of the outlet</td>
<td>10.9</td>
<td>mm</td>
</tr>
<tr>
<td>7.</td>
<td>nozzle diameter</td>
<td>25</td>
<td>mm</td>
</tr>
</tbody>
</table>

#### Figure 2. Shows the CAD geometry of the Nozzle

#### 3.2. Selection of Material

In this work brass material was utilized for the construction of the nozzle. The composition of the brass is shown in the Table 3.

#### Table 3 : Material Composition

<table>
<thead>
<tr>
<th>Sl No</th>
<th>Component</th>
<th>WT%</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>Carbon</td>
<td>60-63</td>
</tr>
<tr>
<td>2.</td>
<td>Ferrous</td>
<td>Max 0.35</td>
</tr>
<tr>
<td>3.</td>
<td>Others</td>
<td>Max 0.5</td>
</tr>
<tr>
<td>4.</td>
<td>Lead</td>
<td>2.5-3.7</td>
</tr>
<tr>
<td>5.</td>
<td>Zinc</td>
<td>35.5</td>
</tr>
</tbody>
</table>

Table 4 illustrates the mechanical properties of the brass which is taken from the materials library. Further, the Table 5 gives the thermal properties of the same material.

#### Table 4: mechanical properties of the Brass

<table>
<thead>
<tr>
<th>SI No</th>
<th>Properties</th>
<th>Value</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>Tensile strength</td>
<td>338-469</td>
<td>MPa</td>
</tr>
<tr>
<td>2.</td>
<td>Yield strength</td>
<td>124-310</td>
<td>MPa</td>
</tr>
<tr>
<td>3.</td>
<td>Elongation at Break</td>
<td>53</td>
<td>%</td>
</tr>
</tbody>
</table>
4. Modulus of Elasticity  |  97  |  GPa  
5. Bulk Modulus        |  140 |  GPa  
6. Poisson’s Ratio     |  0.31|  ---- 
7. Machinability       |  100 |  %    
8. Shear Modulus       |  37  |  GPa  
9. Density             |  8.49|  g/cc 
10. Weight             |  156 |  grams

Table 5: Thermal properties of the brass

<table>
<thead>
<tr>
<th>SI No</th>
<th>Properties</th>
<th>Value</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>Thermal Conductivity</td>
<td>115</td>
<td>W/m-K</td>
</tr>
<tr>
<td>2.</td>
<td>Melting point</td>
<td>885-900</td>
<td>°C</td>
</tr>
<tr>
<td>3.</td>
<td>Solidus</td>
<td>885</td>
<td>°C</td>
</tr>
<tr>
<td>4.</td>
<td>Liquids</td>
<td>900</td>
<td>°C</td>
</tr>
</tbody>
</table>

4. CFD ANALYSIS OF THE NOZZLE

This section describes the considerations made while designing the nozzle. Further, this section gives the outline of the flow at different directions through the nozzle. The results are in analytical in nature using the isentropic flow equations incorporating pressure, temperature, densities at different parameters. The steps involving in the CFD analysis are given in section 3.1.

4.1. Process of CFD Analysis

- Creating Geometry Model
- Generate Mesh
- Specify the boundary conditions
- Specify the initial conditions.
- Setup the CFD simulation
- Result and analysis of simulation.

4.1.1. Geometry Model

This section describes the CAD geometry and the dimensions utilized for construction of the nozzle. The 2-Dimensional geometry was constructed in the sketch as show in figure 4.

4.1.2. Generate Mesh

Fine meshing and mapped mesh has been carried out by utilizing various mesh tools on various section of the nozzle such as at inlet and outlet. The mesh refinement has also been carried out by providing all the boundary conditions as shown in figure 4.

4.1.3. Specify The Boundary Conditions

Figure 5 show the boundary condition utilized for specifying inlet and outlet considerations of the nozzle in CFD simulation.
4.1.4. Setup The CFD Simulation

In this simulation work water is used as fluid. The nozzle material was selected as brass alloy from the material library. The inlet and outlet boundary conditions were applied. Inlet velocity was taken as the boundary condition for the Intel of the nozzle. The inlet velocity was taken as 1.37 m/sec which were practically calculated are presented in section 4.7. Hybrid initialization was employed with 120 iteration was given to solve the CFD analysis. Figure 6 shown the boundary conditions versus number of iterations.

5. RESULTS

5.1. Static Pressure Analysis

Figure 7 illustrate the CFD flow analysis of the fluid in the nozzle. In this analysis it was observed that the static pressure at inlet is maximum which is of about 1.84 bar while at outlet of the nozzle the static pressure is dropping down to 0.0297 Bar.

5.2. Dynamic Pressure Analysis

The reverse has been observed in case of dynamic pressure analysis. It was observed that the inlet pressure was minimum and outlet pressure was maximum. From the Figure 8 the Dynamic pressure at inlet was about 1.74 Bar and for the outlet dynamic pressure was reported to be 2.08 Bar. The increase in the pressure was attributed due to decrease in the cross-sectional area of the nozzle at the outlet.

5.3. Density

Figure 9 show the density plot of the fluid in the nozzle. It was observed that the density was uniform in every section of the nozzle.
5.4. Velocity analysis

Figure 10 shows the velocity plot in the nozzle at inlet which was reported to be 1.29 m/sec while the outlet Velocity reported to be 6.45 m/sec. It was observed that there was about 5 time increase in velocity at the nozzle outlet.

5.5. Steam Function

Figure 11: shows steam function in the developed nozzle. It was observed that maximum and minimum Steam function in nozzle was 6.63 kg/sec and 13.265 kg/sec respectively. Table 6 show the complete CFD analysis result.
The material used to fabricated the nozzle was brass as it is capable of withstanding high pressure and wear resistance. The design consideration were chosen as per the results got in the CDF analysis. The final product of the nozzle is shown in the Figure 13.

Sequence of operations done in the lathe machine.
1. Turning.
2. Facing.
3. Tapering.
4. Drilling.
5. Inside tapering.
7. Threading cutting

6. EXPERIMENTAL CALCULATIONS

6.1. Practical Calculation

Equation 3 is used for Free fall fluid from a define height without application of the nozzle. This equation is used to calculate the time required for the fluid to reach the ground. From this the exit velocity of the fluids is calculated. The observed Fluid flow in x-direction X is 0.6 m and the nozzle position height y-direction Y is 0.9 m.

\[ H = \frac{1}{2} gt^2 \]  \hspace{1cm} (3)

\[ 0.9 = \frac{1}{2} \times 9.81 \times t^2 \]
\[ t = 0.437s \]
\[ X = v_1 \times t \]
\[ 0.6 = v_1 \times 0.437 \]

\[ V_1 = 1.37 \text{m/sec} \]

The fluid flow practical analysis by considering the nozzle is given below. After placing the nozzle to the flow of the fluid the distance covered in the X-direction has been increased to 6.2 m while the height has been increased to 1.211 m. from this the velocity with placing the nozzle is calculated. The final results were tabulated in the Table 7.

\[ X = 6.2m \]
\[ Y = 1.211m \]

\[ H = \frac{1}{2} gt^2 \]

\[ 1.211 = \frac{1}{2} \times 9.81 \times t^2 \]
\[ t = 1 \]
\[ X = v_2 \times t \]
\[ 6.2 = v_2 \times 1 \]

\[ V_2 = 6.2 \text{m/sec} \]

<table>
<thead>
<tr>
<th>S.I NO</th>
<th>PRESSURE (Bar)</th>
<th>VELOCITY (m/sec)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>P1 2.58</td>
<td>V1 1.37</td>
</tr>
<tr>
<td></td>
<td>P2 1.96</td>
<td>V2 6.2</td>
</tr>
</tbody>
</table>

Table 7: Practical velocity calculation with and without placing the nozzle \( V_1, v_2 \) respectively.

7. THEORICAL CALCULATIONS

7.1. By Using Bernoulli’s Equation

From the Bernoulli’s Equation
\[ \frac{p_1}{pg} \frac{v_1^2}{2g} + z_1 = 0 \]

Let us consider P1 and P2 Pressure of water at inlet and outlet respectively which are taken from experimental values and V1 and V2 are determined.

\[ Z_1 = \text{constant} \]
\[ \frac{2.58 \times 10^3}{1000 \times 9.81} + \frac{v_1^2}{2 \times 9.81} = 0 \]

\[ V_1 = 2.271 \text{ m/sec} \]

\[ \frac{p_2}{pg} \frac{v_2^2}{2g} + z_2 = 0 \]

\[ Z_2 = \text{constant} \]
\[ \frac{1.96 \times 10^4}{1000 \times 9.81} + \frac{v_1^2}{2 \times 9.81} = 0 \]

\[ V_2 = 6.261 \text{ m/sec} \]

<table>
<thead>
<tr>
<th>S.I No</th>
<th>PRESSURE (Bar)</th>
<th>VELOCITY (m/sec)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>P1</td>
<td>V1</td>
</tr>
<tr>
<td></td>
<td>2.58</td>
<td>2.271</td>
</tr>
<tr>
<td>2</td>
<td>P2</td>
<td>V2</td>
</tr>
<tr>
<td></td>
<td>1.96</td>
<td>6.261</td>
</tr>
</tbody>
</table>

Comparison of overall results on analysis, practical and theoretical as shown in below Table 8. It was observed that the practical and CDF analysis values are in close proximity as shown in figure 14.

Table 8: Shows the results of the nozzle in analysis, practical and theoretical

<table>
<thead>
<tr>
<th>SI No</th>
<th>Type of Result</th>
<th>Pressure(bar)</th>
<th>Velocity(m/sec)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Analysis</td>
<td>P1</td>
<td>1.74</td>
</tr>
<tr>
<td></td>
<td></td>
<td>P2</td>
<td>2.08</td>
</tr>
<tr>
<td>2</td>
<td>Theoretical</td>
<td>2.58</td>
<td>2.271</td>
</tr>
<tr>
<td></td>
<td></td>
<td>V1</td>
<td>1.29</td>
</tr>
<tr>
<td>3</td>
<td>Practical</td>
<td>2.58</td>
<td>1.37</td>
</tr>
<tr>
<td></td>
<td></td>
<td>V2</td>
<td>6.45</td>
</tr>
<tr>
<td></td>
<td></td>
<td>P1</td>
<td>2.58</td>
</tr>
<tr>
<td></td>
<td></td>
<td>P2</td>
<td>1.96</td>
</tr>
<tr>
<td></td>
<td></td>
<td>V1</td>
<td>2.271</td>
</tr>
<tr>
<td></td>
<td></td>
<td>V2</td>
<td>6.261</td>
</tr>
</tbody>
</table>

Figure 14. Showing the bar graphs of the results.

8. CONCLUSION

A Nozzle is fabricated to determine the pressures and velocities at inlet and outlet respectively. Validated with good accordance with the analytical, practical and theoretical results found are all properties approximately get same value.

REFERENCES


