STUDY OF NOZZLE INJECTION PERFORMANCE USING CFD

Damora Rhakasywi
Mechanical Engineering
Universitas Pembangunan Nasional ”Veteran” Jakarta
Jakarta Selatan, Indonesia
rhakasywi@upnvj.ac.id

Albi Hagi Rahmat
Mechanical Engineering
Universitas Pembangunan Nasional ”Veteran” Jakarta
Jakarta Selatan, Indonesia

Marcellino Rayhan Irwandi
Mechanical Engineering
Universitas Pembangunan Nasional ”Veteran” Jakarta
Jakarta Selatan, Indonesia

***Abstract*—In the current era of modern technology, a deep understanding of fluid flow and computational fluid dynamics (CFD) is crucial in developing and improving the design of engineering equipment, including nozzles. In this research, we apply CFD analysis to study the influence of nozzle design on injection molding nozzle performance. Our main goal is to understand and predict the fluid flow behavior inside the nozzle and can analyze flow characteristics such as speed, pressure. The nozzle geometry used is the inlet nozzle diameter of 35 mm and the outlet diameter of 3 mm. The meshing used is 70,578 elements. This research uses 2 fluid variable parameters, namely, water and polypropylene. pressure contour with fluid parameters, namely water with a maximum pressure value of 1.57 ×10^5 Pascal. At outlets with water fluid parameters, the velocity around the outlet ranges from 1.76 m/s to 1.76 ×10 m/s. Meanwhile, the polypropylene fluid parameters range from 1.74 m/s to 1.74 ×10 m/s. At the nozzle outlet, the flow velocity increases from the outer side to the middle side of the fluid flow around the nozzle outlet.**

***Keywords*—nozzle,CFD,injection,fluids,velocity.**

I. Introduction

In the current era of modern technology, a deep understanding of fluid flow and computational fluid dynamics (CFD) has become crucial in developing and improving the design of engineering equipment, including nozzles. Nozzles play an important role in a variety of industrial and engineering applications, from jet engines to combustion and spraying. Nozzle performance directly affects the efficiency and effectiveness of the existing system. Therefore, understanding and optimizing fluid flow through the nozzle becomes very important. CFD analysis allows engineers to understand the fluid flow behavior around a nozzle in detail and accurately without having to rely on expensive and complicated physical tests.

Injection molding technology has become the main method for manufacturing plastic components on a large scale. One of the critical components in this process is the nozzle, which functions to direct the liquid material into the mold. However, optimal nozzle design and operation is often a challenge.

Computational fluid dynamics (CFD) analysis has proven to be an effective tool in understanding and optimizing processes such as these. Using CFD, we can simulate the flow of liquid material through a nozzle and predict important variables such as pressure, speed and temperature.

In this research, we apply CFD analysis to study the influence of nozzle design on injection molding nozzle performance. Our main goal is to understand and predict fluid flow behavior inside the nozzle and be able to analyze flow characteristics such as speed, pressure, as well as identify design parameters that can optimize material flow, reduce product defects, and increase energy efficiency.

We hope that this research will provide new insights for engineers and designers in the industry, and help them make better and more informed design decisions.

This research aims to study the flow pattern in the injection molding nozzle used in the tool made in the Capstone Design course, namely the Polymer Injection Molding tool. This journal

focuses on understanding changes in speed, pressure and distribution of fluid flow around the nozzle. By utilizing a numerical simulation approach, this research is expected to provide deeper insight into the performance of the nozzle on this tool.

II. Methods

The research method used in this research is the Computational Fluid Dynamics (CFD) method using ANSYS Fluent Student Version software. This method is divided into three stages, namely: pre-processing stage, (processing) stage, and post-processing stage. The geometry used in this experiment was created using the Autodesk Inventor software. The geometry that has been created is then exported to the Ansys Workbench software.

A. Object

The object of this research we use this time is the nozzle that is used in the Capstone Design project, namely Vertical Polymer Injection Molding. The nozzle is made of aluminum. The nozzle functions to supply liquid polymer material into the mold.

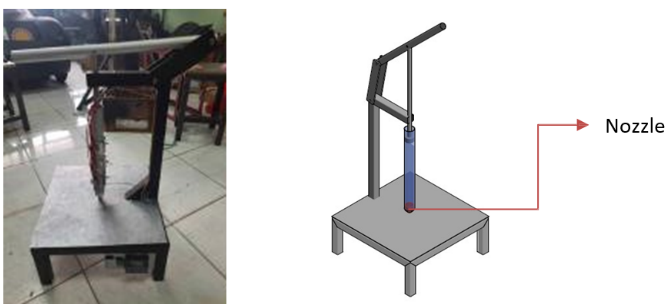


Fig. 1: Vertical Polymer Injection Molding.

B. Geometri Nozzle

The three-dimensional propeller model was processed using Autodesk Inventor and reprocessed in the Ansys program to create the fluid domain geometry. The nozzle inlet diameter is 35 mm and the outlet diameter is 3 mm.

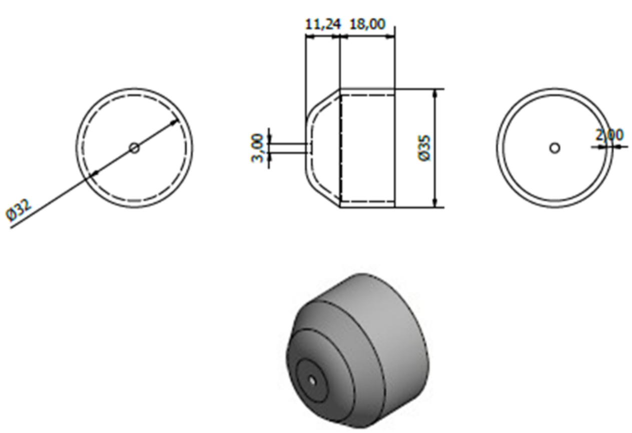


Fig. 2: Iteration of Computer Fan Simulation.

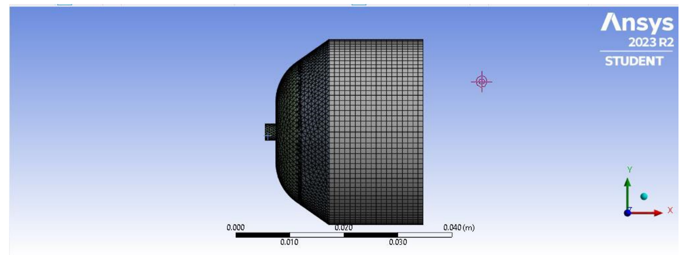


Fig. 3: Mesh in Nozzle.

C. Mesh of Geometry

In order to create the meshing of a geometry, we need to switch the workbench from geometry to meshing. After creating the geometry, it is required to divide the control volume into a smaller number of Nodes and element of finite size, therefore it is called a finite volume method. The method of splitting the Control volume into small finite size volume is known as a meshing of the control volume.

TABLE 1: Specification of Mesh

No.	Parameter	Size/Specification
1.	Element size	0.001
2.	Maximum size	0.002
3.	Node	43419
4.	Element	70578

D. Solving

1) Governing Equation: The governing equations used by the CFD software package for this study are as follows: Conservation of Mass (Continuity Equation):

$$\frac{D_\rho}{D_t} = \rho.\nabla.V_x = 0 \tag{1}$$

2) Boundary Condition:

1) Inlet
2) Wall
3) Outlet

Specification of the boundary zones has to be done in WORKBENCH only, as there is no possibility to specify the boundary zones in FLUENT. Therefore proper care has to be taken while defining the boundary conditions in WORKBENCH. With all the zones defined properly the mesh is exported to the solver. The solver used in this problem is ANSYS FLUENT. The exported mesh file is read in Fluent for solving the problem.

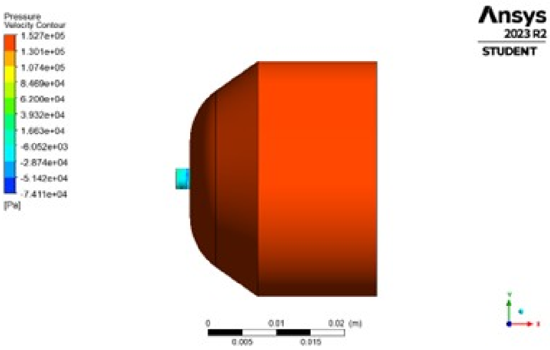
TABLE 2: Parameter properties

	Properties	
	Water	Polipropilen
Viscocity (kg/m.s)	0.001003	0.01
Density (kg/m^3)	998.2	9100

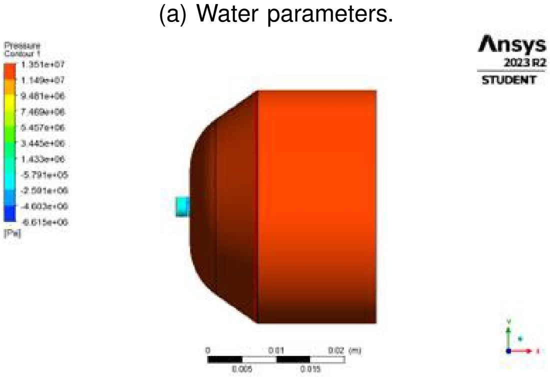
III. Results

A. Contours Pressure on the Nozzle

In figure 4 is the pressure contour of the fluid flow in the nozzle in a 3-dimensional view which displays the flow in the nozzle. Figure 4a is a pressure contour with fluid parameters, namely water,with a maximum pressure value of 1.57 × 10^5 Pascal and a minimum of -7.41 × 10^4 Pascal. Meanwhile, Figure 4b is a pressure contour with fluid parameters, namely Polypropylene with a maximum pressure value of 1.35 × 10^7 Pascal and a minimum of-6.61 × 10^6 Pascal.



(a) Water parameters.



(b) Polypropylene parameters.

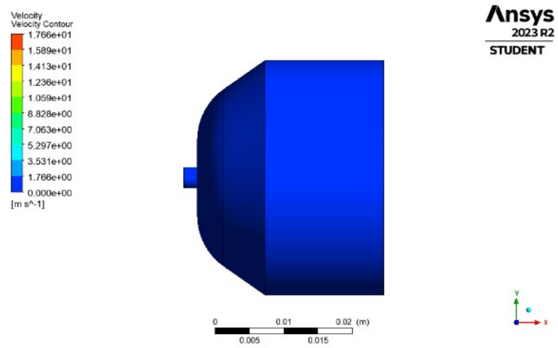
Fig. 4: Pressure contour at the nozzle.

B. Velocity Contour on the Nozzle

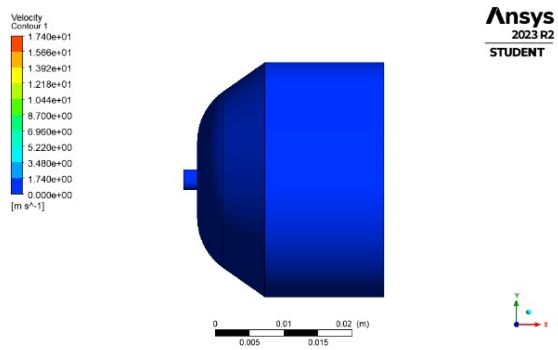
In figure 5 is the contour of the fluid flow velocity in the nozzle in a 3-dimensional view which displays the flow in the nozzle. Figure 5 (a) is a velocity contour with fluid parameters, namely water with a maximum pressure value of 1.76 ×10 m/s. Meanwhile, Figure 5 (b) is a pressure contour with fluid parameters, namely polypropylene with a maximum pressure value of 1.74×10 m/s.

C. Nozzle Pressure Inlet and Outlet Contours

In figure 6 and 7. is a view of each inlet and outlet pressure contour on the nozzle. It can be seen that there is a change in the value and color of the contour and a decrease in pressure.

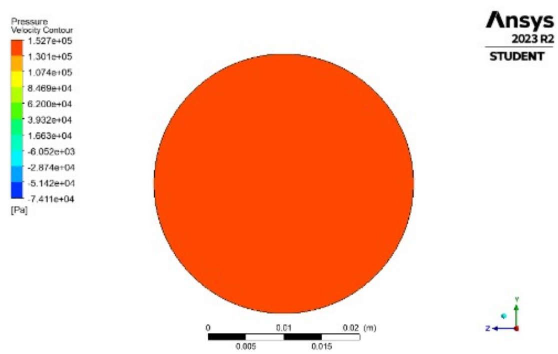


(a) Water parameters.

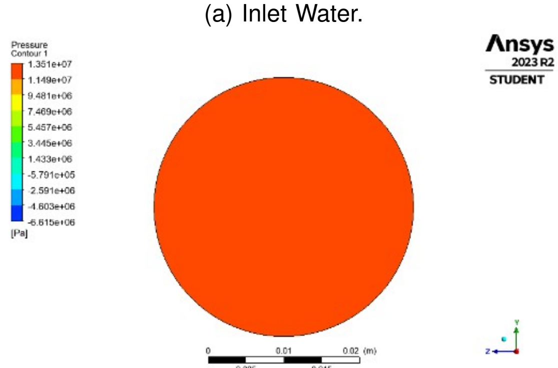


(b) Polypropylene parameters.

Fig. 5: Velocity contour at the nozzle.

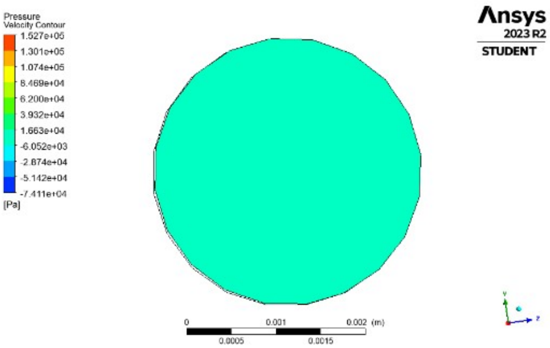


(a) Inlet Water.

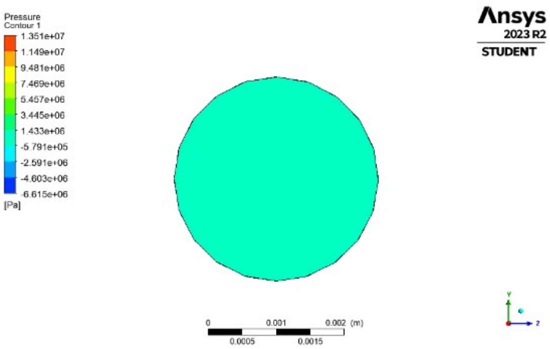


(b) Inlet Polypropylene.

Fig. 6: Inlet Pressure contour at the nozzle.

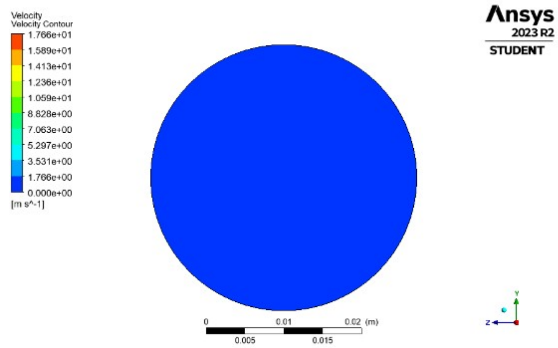


(a) Outlet Water.

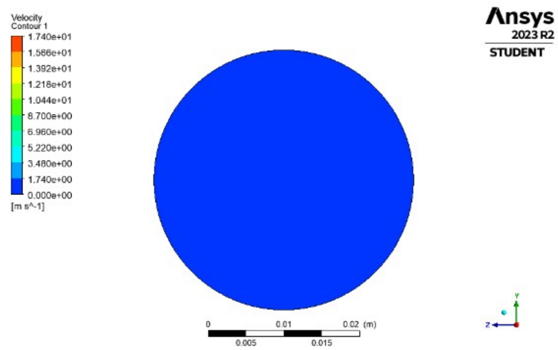


(b) Outlet Polypropylene.

Fig. 7: Outlet Pressure contour at the nozzle.



(a) Inlet Water.



(b) Inlet Polypropylene.

Fig. 8: Inlet Velocity contour at the nozzle.

D. Nozzle Velocity Inlet and Outlet Contours

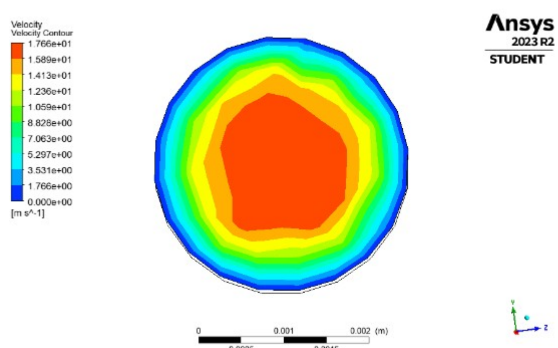
In figure 8 and figure 9 is a view of each inlet and outlet velocity contour on the nozzle. It can be seen that at the outlet there is a change in the velocity value in the fluid flow around the outlet. At outlets with water fluid parameters, the velocity around the outlet ranges from 1.76 m/s to 1.76 ×10 m/s. Meanwhile, the polypropylene fluid parameters range from 1.74 m/s to 1.74 ×10 m/s.

E. Velocity Streamline

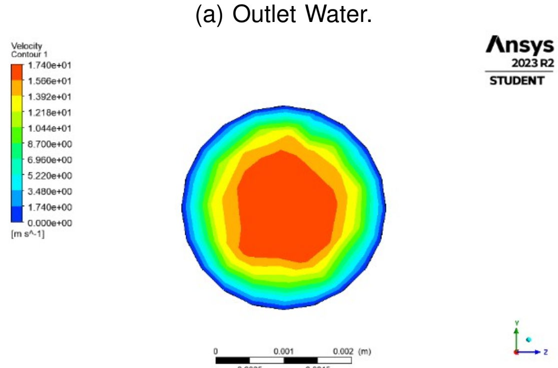
In the figure 10 you can see a visualization of the fluid flow that occurs in the nozzle. Describes fluid moving from the inlet to the outlet.

F. Flow Velocity at the Nozzle

The velocity magnitude graph in Figure 11 shows the movement of a particle from positions 0 to - 0.03 m. The particle moves from right to left, and its speed decreases as it moves. At the initial position the magnitude velocity tends to be stable, but at some point the magnitude velocity increases and then decreases again.



(a) Outlet Water.



(b) Outlet Polypropylene.

Fig. 9: Outlet Velocity contour at the nozzle.

IV. Conclusions

The main objective of this article is to understand and predict the behavior of fluid flow in the nozzle and be able to analyze flow characteristics such as speed, pressure

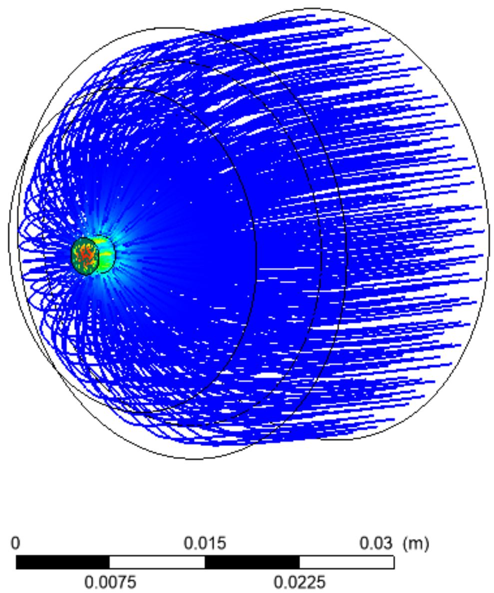


Fig. 10: Velocity Streamline.

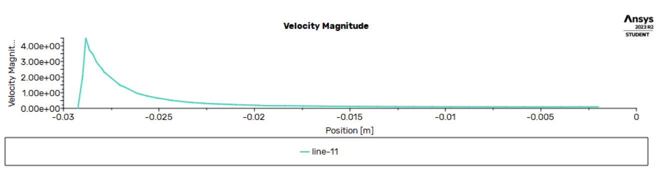


Fig. 11: Graph of Velocity Magnitude.

in the nozzle injector. From the simulation results we can find out :

1) In the simulation with polypropylene fluid parameters, it shows a pressure value that is greater than water fluid due to differences in density and viscosity. Namely with a maximum pressure value of 1.35×10^7 Pascal.
2) In the speed contour there is no significant difference in these two parameters.
3) From the simulation results, it can be seen that there is an increase in the speed and pressure of the fluid flow around the nozzle outlet. At the nozzle outlet, the flow velocity increases from the outer side to the middle side of the fluid flow around the nozzle outlet.

References

[1] Pathak V, Gupta S. Study of Nozzle Injector Performance Using CFD. Int J Recent Adv Mech Eng. 2015;4:153-160. doi: 10.14810/ijmech.2015.4312.

[2] Jagtap R. Theoretical & CFD analysis of de Laval nozzle. Int J Mech Prod Eng. 2014;2:33- 36.

[3] Piscaglia F, Giussani F, Hèlie J, Lamarque N, Aithal SM. Vortex Flow and Cavitation in Liquid Injection: A Comparison between High-Fidelity CFD Simulations and Experimental Visualizations on Transparent Nozzle Replicas. Int J Multiphase Flow. 2021;138:103605. doi: 10.1016/j.ijmultiphaseflow.2021.103605.

[4] Ramesh Kumar R, Devarajan Y. CFD simulation analysis of two-dimensional convergent- divergent nozzle. Int J Ambient Energy. 2020;41(13):1505-1515.

[5] Pathan KA, Khan SA, Dabeer PS. CFD analysis of effect of Mach number, area ratio and nozzle pressure ratio on velocity for suddenly expanded flows. In: 2017 2nd International Conference for Convergence in Technology (I2CT). IEEE; 2017. p. 1104-1110.

[6] De Gaetano F, Serrani M, Brubert J, Stasiak J, Moggridge G, Costantino ML. Injection moulding process: CFD evaluation on the orientation of polymeric chains for manufacturing heart valves. Int J Artif Organs. 2015:389-389.

[7] Jalaluddin J, Akmal S, Nasrul ZA, Ishak I. Analisa Profil Aliran Fluida Cair dan Pressure Drop pada Pipa L menggunakan Metode Simulasi Computational Fluid Dynamic (CFD). Jurnal Teknologi Kimia Unimal. 2019;8(1):97-108.