Advanced and Sustainable Technologies
ASET
E-ISSN 2976-2294

Volume 3, No 2, December 2024 [61-70]

Effect of Inlet-Outlet Diameter on the Water Flow in Gully Trap for Water Drainage System: CFD Analysis

Muhammad Syafiq Bahanuddin[1*]

[1]Faculty of Mechanical Engineering & Technology, Universiti Malaysia Perlis (UniMAP), 02600 Arau, Perlis, Malaysia

Received 14 August 2024, Revised 5 September 2024, Accepted 10 September 2024

ABSTRACT

The efficient functioning of bathroom drainage systems is critical for maintaining hygienic and odor-free living spaces. Central to these systems is the gully trap, which plays a key role in preventing the escape of sewer gases while allowing wastewater to flow freely. This study employs Computational Fluid Dynamics (CFD) to analyze the internal flow characteristics within a gully trap, focusing on the impact of varying inlet and outlet diameters on velocity and pressure distribution. Using the SimFlow software, simulations were conducted to visualize and quantify the flow behavior within the trap. The study involved generating an unstructured mesh with a maximum node limit of 200,000 to ensure a detailed capture of flow dynamics. The finite volume method (FVM) and appropriate turbulence models, such as k-ε or k-ω, were utilized for accurate simulations. Verification through grid independence tests and sensitivity analyses ensured the reliability of the results. The findings revealed significant velocity and pressure distribution variations based on changes in inlet and outlet diameters, providing insights into optimizing gully trap design for enhanced performance. This study contributes to environmental engineering by offering a deeper understanding of fluid dynamics in drainage systems, paving the way for developing more efficient and reliable gully traps. The insights gained are expected to aid in reducing blockages and improving water flow efficiency, ultimately enhancing the sustainability and effectiveness of plumbing systems.

Keywords: Gully trap, Inlet-Outlet diameter, Computational fluid dynamics, Simflow 4.0

1. INTRODUCTION

Bathroom drainage systems are crucial in maintaining hygiene and comfort in residential and commercial buildings [1]. These systems are designed to remove wastewater [2] from bathrooms efficiently, preventing potential health hazards [3] and structural damage [4] caused by water accumulation. One of the primary functions of a bathroom drainage system is to prevent the backflow of sewer gases, which can carry unpleasant odors and harmful microorganisms into living spaces. The gully trap is a key component in achieving this, specifically engineered to create a water seal that blocks sewer gases [5] while allowing wastewater to pass through without obstruction. The gully trap's design is fundamental to its functionality, as it relies on specific fluid dynamics principles to maintain its effectiveness. The wastewater flow through the gully trap must be carefully managed to ensure the water seal is always maintained. This involves controlling the velocity and pressure of the water entering the trap to prevent siphoning or loss of the water seal, which could release foul odors.

---

*Corresponding author: syafiqbaha99@gmail.com

Fluid dynamics in confined spaces such as gully traps [6] can be complex due to the interactions between the water flow and the trap geometry. Understanding the internal flow characteristics, such as the velocity and pressure distributions within the gully trap, is essential for optimizing its design. These factors directly impact the trap's ability to prevent odor escape while efficiently channeling wastewater. By studying these dynamics, engineers can improve the performance and durability of gully traps, ensuring they function effectively under various conditions and contribute to the overall reliability of bathroom drainage systems. Computational Fluid Dynamics (CFD) is a powerful tool that enables detailed analysis and visualization of these interactions. By simulating the internal flow, CFD can provide insights into the behavior of fluids under various conditions, leading to better design and performance of gully traps.

Rietveld et al. [7] studied how urban runoff carries solids from street surfaces into drainage systems, reducing their hydraulic capacity through sedimentation and impacting water quality due to pollutant discharge. To mitigate these effects, gully pots are fitted with sand traps that act as settling tanks to remove suspended solids. This study uses Particle Image Velocimetry (PIV) and Laser Doppler Anemometry (LDA) to analyze how factors like gully pot geometry, discharge rates, sand trap depth, and sediment bed levels affect flow dynamics and sedimentation processes. They revealed that sediment bed morphology significantly influences flow patterns and removal efficiency, challenging the notion of gully pots as completely mixed reactors and highlighting the role of turbulence and bed levels in resuspension and erosion. Lo et al. [8] explored the miniaturization of gullies in building drainage systems by adding a streamlined bump to maintain a minimum water trap height. Experimental and numerical analyses, including Computational Fluid Dynamics (CFD), were conducted to examine the hydrodynamic behavior of air-water flows, glass ball transport, and self-purification capabilities in different gully designs. Their results showed that gullies with beveled nozzles perform better in transporting glass balls due to lower pressure gradients and reduced air entrainment, enhancing flow rates and self-cleaning efficiency. Besides, Jean and Gormley [9] focused on improving the modeling of water trap seals in sanitation systems to predict better their response to air pressure transients, a crucial aspect for maintaining the safe removal of human waste. Using computational fluid dynamics (CFD) and the AIRNET model, the research highlights the impact of transient air pressure frequencies on the performance of water traps and identifies vulnerabilities based on device geometry. Developing a dynamic velocity decrement model and a frequency-dependent internal energy term enhances the predictive accuracy of water trap behavior under varying pressure conditions, particularly for frequencies between 1 Hz and 8 Hz.

Gormley et al. [10] provided evidence that pathogens can become aerosolized and transported through sanitary plumbing systems, potentially entering buildings via empty U-traps. Experiments with a model plumbing system demonstrated that toilet flushes create turbulence sufficient to aerosolize pathogens, which can then travel between floors on typical airflows within the system. They suggest that empty U-traps, especially in buildings with high pathogen loads like hospitals, pose a significant risk for cross-transmission of airborne pathogens, highlighting the need for greater attention to this transmission mode. Lo et al. [11] investigated the hydrodynamic characteristics of air-water flows in multi-entry gullies used in building drainage systems, focusing on the effects of swirl generation vanes (SGV). Using experimental methods and numerical simulations with Flow-3D, the research examines air entrainment, airflow pressure variations, bubble trajectories, and self-cleaning processes in gullies with and without SGV. The results show that SGVs help stabilize air-bubble interactions and increase discharge rates by over 7%, reducing the self-purification period and improving the overall performance of the drainage system. Ardejani et al. [12] highlighted the flexibility of the finite volume method (FVM) in handling complex geometries and ensuring mass, momentum, and energy conservation. This is crucial for accurately modeling drainage systems, including gully traps, to predict how changes in design parameters such as inlet and outlet diameters affect overall performance.

This study uses computational fluid dynamics (CFD) to enhance the design and performance of gully traps, which are crucial components in drainage systems that prevent sewer gases from entering living spaces while allowing wastewater to flow efficiently. The simulation model of the gully traps was meticulously developed in SolidWorks, a robust CAD software, and exported in STL format to ensure high precision and compatibility. This 3D model was then imported into SimFlow 4.0 [13], a powerful CFD software, where the simulation settings were configured to replicate real-world conditions. The results were visualized to provide a comprehensive understanding of the flow behaviors. By applying CFD, engineers are able to analyze complex fluid dynamics within the gully traps, including factors like velocity distributions, pressure variations, and turbulence patterns. This detailed analysis identifies potential design flaws and optimization opportunities, such as improving the trap geometry to enhance water flow efficiency or adjusting the trap's dimensions to block sewer gases better. The advanced numerical methods employed in CFD simulations allow for the prediction of how different design modifications will perform under various operating conditions, thus facilitating the development of more efficient, durable, and reliable drainage systems. This approach improves the functionality and effectiveness of gully traps. It contributes to public health and safety by minimizing the risks associated with sewer gas leaks and ensuring efficient wastewater management.

2. MATERIAL AND METHODS

The methodology for analyzing fluid flow within a gully trap involves several key steps, focusing on different inlet and outlet configurations to understand the effects on flow characteristics. Figure 1 shows the three-dimensional model of the gully trap used in the bathroom drainage. This study created and examined various configurations with inlets sized 3 inches (76.2 mm) and 4 inches (101.6 mm), paired with outlet sizes of 2 inches × 2 inches (50.8 mm × 50.8 mm), 2 inches × 3 inches (50.8 mm × 76.2 mm), and 3 inches × 3 inches (76.2 mm × 76.2 mm). The front view of the models is depicted in Figure 2, and the combinations of inlet-outlet diameter are summarized in Table 1. Three designs are obtained from the combinations: Design 1, Design 2, and Design 3.

The CFD simulations are conducted using SimFlow software, which employs the finite volume method (FVM) to solve the fluid flow. Mesh generation is a critical step in the analysis, as it influences the accuracy and convergence of the solution. A combination of structured and unstructured meshes is used to capture the intricate details of the gully trap geometry. The mesh parameters are set with a base type of "box" and 30, 30, and 100 divisions (Figure 3 a), respectively. The material point is set at the fluid location within the inner geometry. The hex-dominant meshed model is shown in Figure 3 (b). Solver settings are configured with the SIMPLE solver [14] type, using the RANS turbulence model with k-ω SST for turbulence modeling [15]. Water is chosen as the fluid property, and the inlet and outlet boundary conditions are set as specified. Monitor sampling is established to track key flow parameters throughout the simulation.

Boundary conditions are crucial for defining the flow characteristics at the inlet and outlet of the gully trap. Figure 4 shows the boundary conditions defined on the fluid domain of the gully trap. A velocity inlet boundary condition is applied with a pressure type set to zero gradient and a velocity type set to surface normal fixed value at 0.5 m/s. Turbulence parameters include a turbulent intensity inlet (k type) with an intensity of 0.05 and a turbulent mixing length inlet (ω type) with a mixing length of 1×10^{-3} m. A pressure outlet boundary condition is used at the outlet, with flow parameters set to fixed value pressure and pressure inlet-outlet velocity for the velocity type. Turbulence parameters at the outlet are set to zero gradient for both k and ω. The mesh elements generated on all models are summarized in Table 2. The mesh qualities of the model, such as skewness and aspect ratio, are acceptable, as indicated by SimFlow software. In summary, this methodology, which includes varying inlet and outlet sizes, a structured approach to mesh generation, and robust numerical methods, ensures a thorough analysis of fluid flow within gully traps. The approach provides valuable insights into the impact of design variations on flow characteristics, aiding in optimizing gully trap designs for improved performance and efficiency.

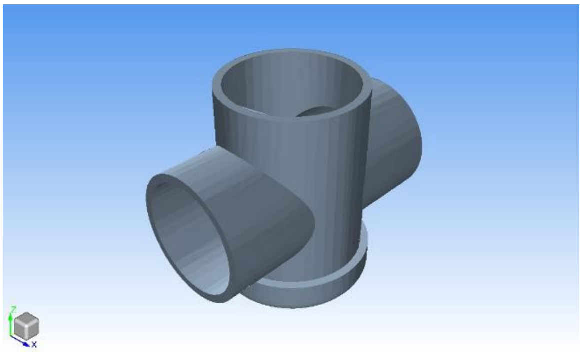


Figure 1: 3D model of gully trap used in the bathroom drainage.

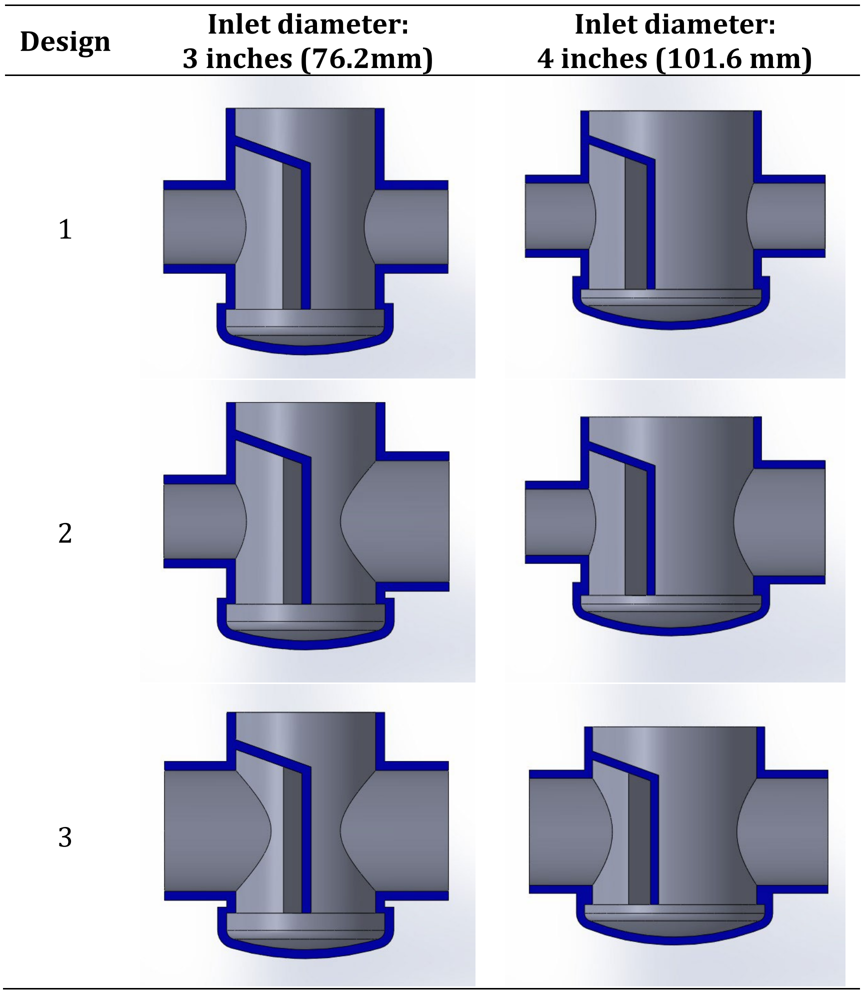
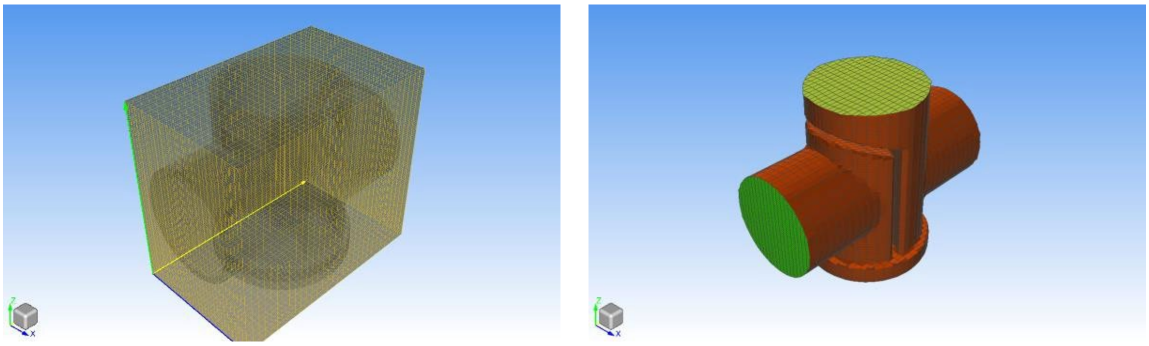


Figure 2: Front view of the model for designs 1-3 with different inlet diameters.

Table 1: Combination of inlet-outlet diameters in Designs 1-3.

Design	Outlet diameter Inch (mm)		Inlet diameter Inch (mm)	
	1	2	1	2
1	2 (50.8 mm)	2 (50.8 mm)	3 (76.2 mm)	4 (101.6 mm)
2	2 (50.8 mm)	3 (76.2 mm)		
3	3 (76.2 mm)	3 (76.2 mm)		



(a)

(b)

Figure 3: (a) Box-type meshing in the SimFlow and (b) Hex-dominant mesh model of the fluid domain.

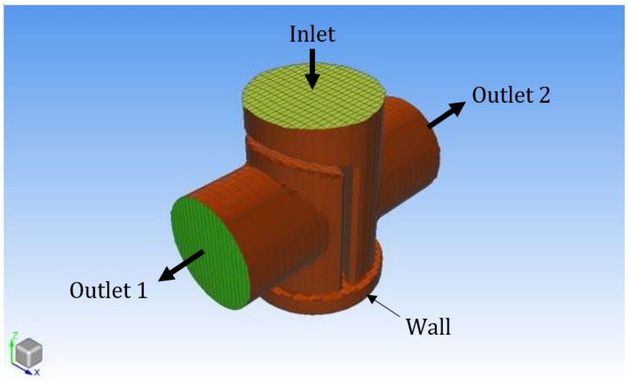


Figure 4: Boundary conditions of the fluid domain.

Table 2: Meshing data for Designs 1-3.

Mesh Parameters	Inlet diameter: 3 inches (76.2mm)			Inlet diameter: 4 inches (101.6 mm)		
	Design 1	Design 2	Design 3	Design 1	Design 2	Design 3
Cells	29466	33751	37982	31496	34475	37204
Nodes	32796	37685	42226	34969	38162	41125
Element type	Hex-dominant	Hex-dominant	Hex-dominant	Hex-dominant	Hex-dominant	Hex-dominant
Max skewness	3.341607	3.341607	3.341607	3.341067	3.341067	3.341067
Max aspect ratio	13.29675	13.29675	13.29675	15.08137	15.08137	15.08137

3. RESULTS AND DISCUSSION

The analysis of fluid flow within the gully trap using CFD simulations has provided critical insights into the behavior of wastewater as it traverses this essential plumbing component. The findings reveal significant variations in pressure and velocity distributions based on different inlet and outlet diameter configurations. Notably, smaller inlet diameters paired with larger outlet diameters result in higher pressure differentials and increased flow velocities. This trend is particularly evident in the configuration with a 3-inch inlet and a 2×2- inch outlet, which exhibited the highest maximum pressure and velocity values. Besides, the simulations demonstrate that increasing the inlet diameter generally reduces maximum pressure and velocity values. This reduction can be attributed to the increased cross-sectional area for the fluid to pass through, thereby reducing flow resistance. Consequently, configurations with larger inlets and outlets showed more moderate pressure and velocity profiles, indicating a more stable and less turbulent flow regime.

Table 3 presents the maximum pressure and velocity observed in different gully trap designs with two varying inlet diameters: 3 inches (76.2 mm) and 4 inches (101.6 mm). The results show that the maximum pressure tends to decrease with an increase in the inlet diameter for all designs. Design 1 significantly reduces maximum pressure from 4.8298 Pa for the 3-inch inlet to 3.4687 Pa for the 4-inch inlet. This trend suggests that a larger inlet diameter helps to reduce pressure buildup within the gully trap. Similarly, the maximum velocity is also lower in gully traps with larger inlets, indicating that the flow rate is more controlled and less turbulent, which could reduce erosion and wear on the trap components. The data indicates that optimizing the inlet diameter is crucial for managing pressure and flow velocity, potentially improving the overall efficiency and longevity of the gully trap designs.

Figure 5 illustrates the distribution of maximum pressure across different gully trap designs. It shows that Design 1 exhibits the highest maximum pressure, particularly for the 3-inch inlet, which aligns with the data presented in Table 3. The result highlights how design variations can significantly impact pressure levels within the gully trap, a critical factor for preventing backflow and maintaining structural integrity. The reduction in maximum pressure for designs with a larger 4-inch inlet further confirms the benefits of increased diameter in mitigating pressure surges within the system.

Figure 6 shows the maximum velocity within the gully trap for different designs. The result demonstrates that maximum velocity is generally higher for designs with smaller 3-inch inlets, particularly in Design 1. This outcome suggests that smaller inlets lead to higher fluid speeds, which could increase the risk of air entrainment and turbulence, potentially affecting the trap's efficiency. Conversely, the lower velocities observed in designs with 4-inch inlets point towards a more stable flow regime, which could be beneficial for minimizing sediment resuspension and ensuring a more effective separation of solids from the wastewater stream.

Table 3: Maximum pressure and velocity of different gully trap designs.

Design	Maximum Pressure (Pa)		Maximum Velocity (m/s)	
	Inlet Diameter		Inlet Diameter	
	3 inches (76.2mm)	4 inches (101.6mm)	3 inches (76.2mm)	4 inches (101.6mm)
1	4.8298	3.4687	2.4327	1.5487
2	1.3598	1.9174	1.4537	1.3077
3	1.285	1.3947	1.5487	1.4197

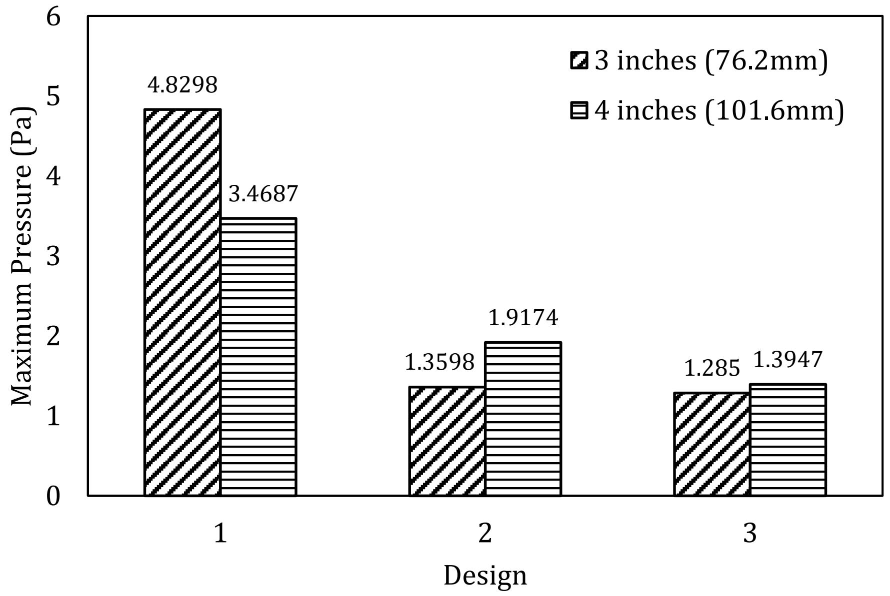


Figure 5: Maximum pressure in the gully trap for different designs.

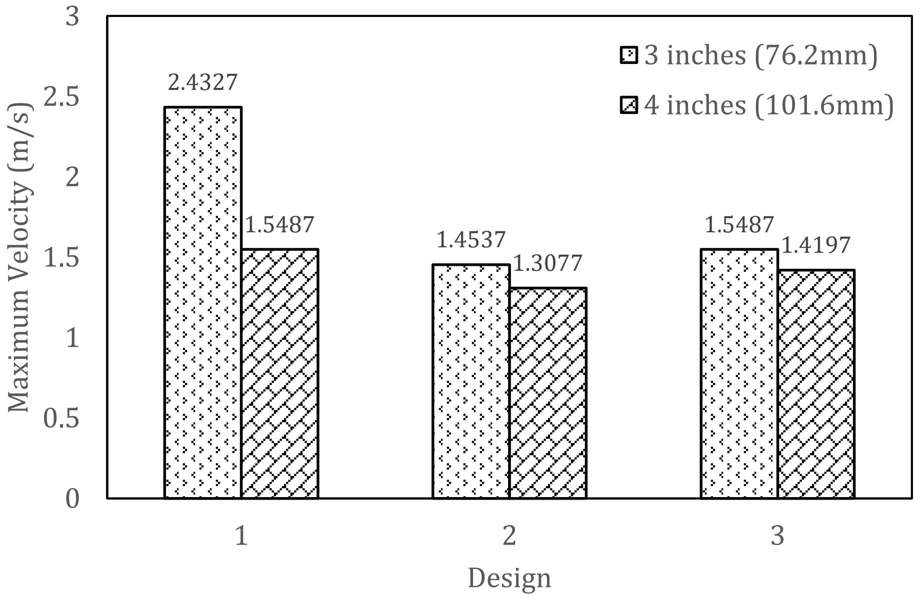


Figure 6: Maximum velocity of the gully trap for different designs.

Figure 7 compares the pressure contours for gully traps with different inlet-outlet diameters. The contour plots visually represent pressure distribution within the traps, revealing high and low-pressure areas that could affect the trap's performance. The comparison shows that larger inlet diameters produce more uniform pressure distributions, reducing localized high-pressure zones that could lead to structural stress or potential failure. This finding emphasizes the importance of designing gully traps with appropriately sized inlets and outlets to optimize pressure management and enhance the durability of the drainage system.

Figure 8 compares velocity contour for different gully trap designs with varying inlet-outlet diameters. The contours highlight the flow patterns and velocity gradients within the gully traps, indicating regions of high velocity that may contribute to increased wear or erosion of the trap surfaces. The results suggest that designs with larger inlet diameters (4 inches) show a more evenly distributed velocity field, which is advantageous for minimizing turbulent flow and preventing sediment agitation. In contrast, smaller inlets (3 inches) tend to create more pronounced velocity gradients, potentially leading to increased turbulence and less efficient sediment settling. Overall, these results emphasize the critical role of inlet-outlet diameter and design considerations in optimizing the hydrodynamic performance of gully traps in building drainage systems. By carefully analyzing pressure and velocity distributions, engineers can make informed decisions to improve these essential components' efficiency, safety, and durability.

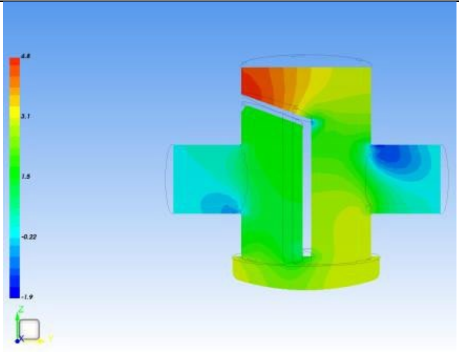
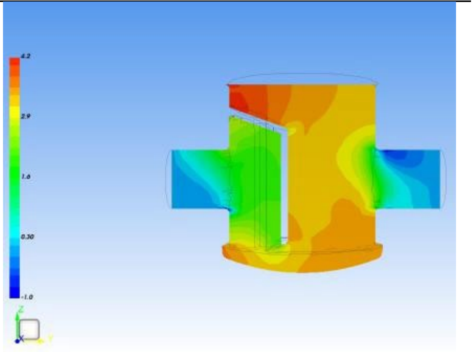
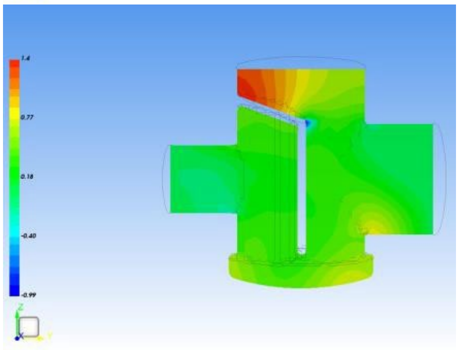
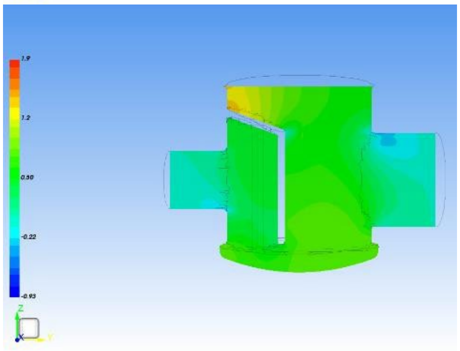
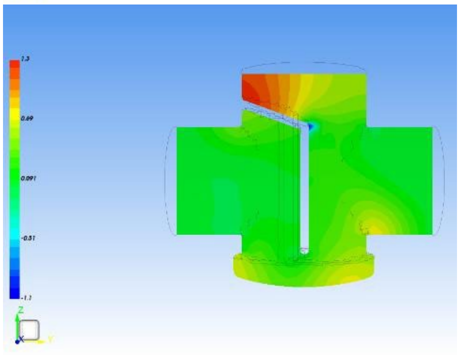
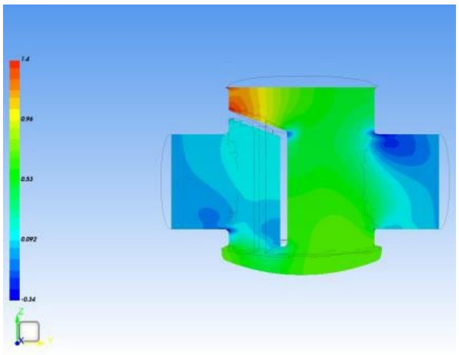
Design	**Inlet Diameter**	
	3 inches (76.2mm)	**4 inches (101.6mm)**
1		
2		
3		

Figure 7: Pressure contour comparison for different inlet-outlet diameters of gully trap.

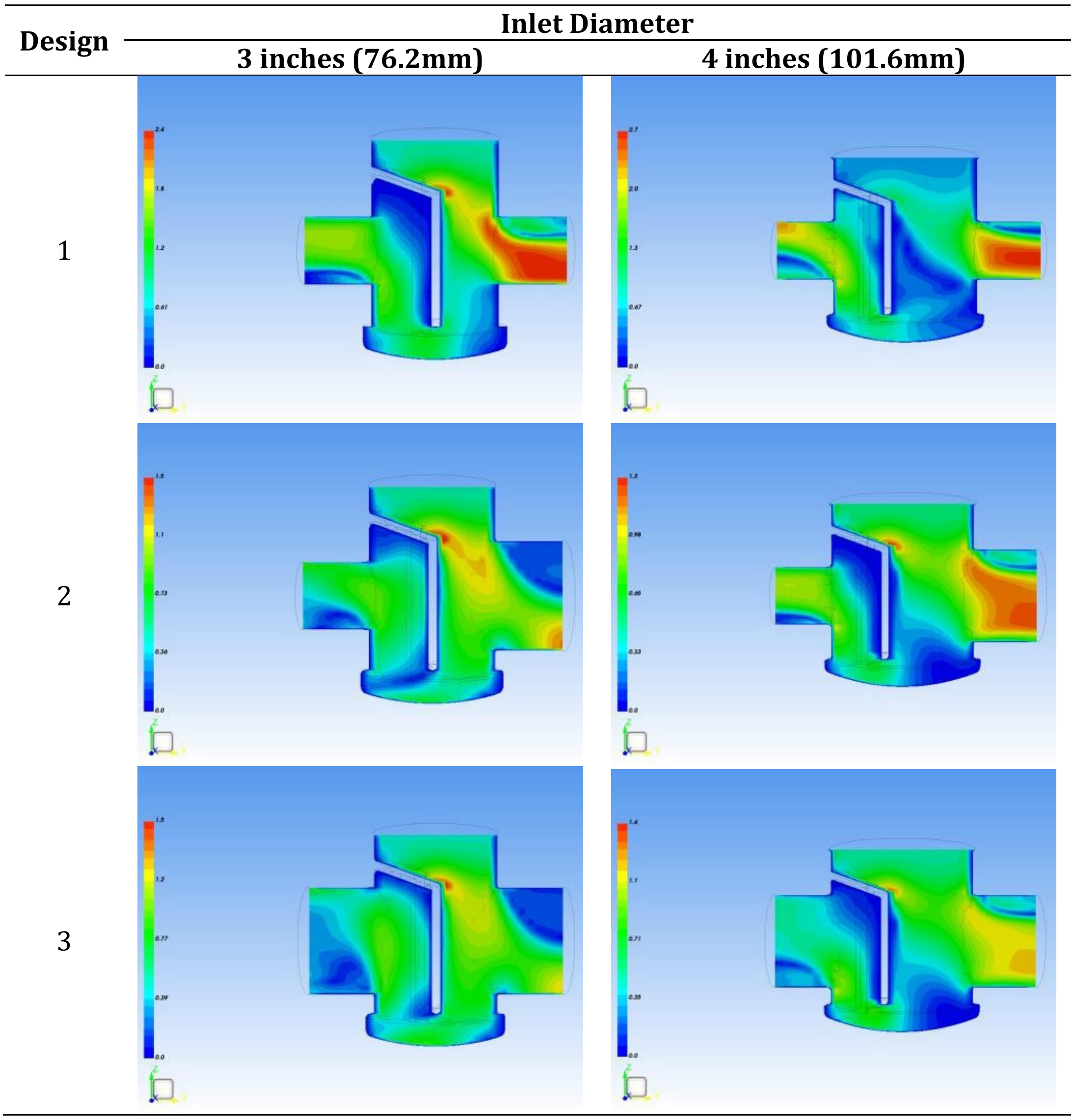


Figure 8: Velocity contour comparison for different inlet-outlet diameters of gully trap.

4. CONCLUSION

This study effectively employed CFD simulations to investigate the flow characteristics within a gully trap, explicitly examining the impact of different inlet and outlet diameters on pressure and velocity distributions. The main findings indicate that smaller inlet diameters combined with larger outlet diameters create higher pressure differentials and increased flow velocities, leading to a more turbulent flow regime. However, larger inlets with well-matched outlet sizes result in more stable and moderate flow characteristics. These insights are crucial for optimizing gully trap designs, as selecting the right inlet and outlet diameters can improve the efficiency and reliability of drainage systems, ensuring effective wastewater management while minimizing the risks of blockages and gas backflow. The application of advanced turbulence models, such as the k-ω SST, demonstrated a high level of precision in capturing the complex fluid dynamics within the gully trap, confirming the accuracy and reliability of the CFD simulations. Overall, this research contributes significantly to environmental engineering by enhancing the understanding of fluid dynamics in drainage systems. The results are expected to guide the development of more efficient and reliable gully traps, thereby improving the sustainability and performance of plumbing systems in modern infrastructure.

ACKNOWLEDGEMENTS

The author deeply appreciates the Faculty of Mechanical Engineering & Technology's crucial support and resources during this research. A special note of thanks goes to the CFD course coordinator for their expert guidance and mentorship.

REFERENCES

[1] Penner, B. Bathroom. Reaktion Books. (2013).
[2] Wilson, A. Drainage and sanitation. In Handbook of ancient water technology, Brill, (2000) pp. 151-179.
[3] Bryson, J., & Battersby, S. Constructions and related matters relevant to environmental health. In Clay's Handbook of Environmental Health, Routledge, (2022) pp. 277-336.
[4] Liao, W. J., Cheng, C. L., & Lee, C. J. Study on the Bathroom Space and the Application of Same-Floor Drainage in Congregate Housing. Water, vol 13, issue 20 (2021) p. 2922.
[5] Coutts, F. J., & Brodie, J. S. Sewer Ventilation and the Intercepting Trap. Journal of the Royal Sanitary Institute, vol 29, issue 4 (1908) pp. 181-201.
[6] Frankl, A., Nyssen, J., Vanmaercke, M., & Poesen, J. Gully prevention and control: Techniques, failures and effectiveness. Earth Surface Processes and Landforms, vol 46, issue 1 (2021) pp. 220-238.
[7] Rietveld, M., de Rijke, D., Langeveld, J., & Clemens, F. Sediment morphology and the flow velocity field in a gully pot: An experimental study. Water, vol 12, issue 10 (2020) p. 2937.
[8] Lo, D. C., Chang, S. W., Liu, H. F., & Chen, C. Y. Experimental and Numerical Study of Hydrodynamic Characteristics of Gullies for Buildings. Water, vol 10, issue 2 (2018) p. 165.
[9] Jean, N. J., & Gormley, M. Modelling water trap seal boundary conditions in building drainage systems: Computational fluid dynamics analysis of unsteady friction to improve accuracy. Building Services Engineering Research and Technology, vol 38, issue 5 (2017) pp. 580-601.
[10] Gormley, M., Aspray, T. J., Kelly, D. A., & Rodriguez-Gil, C. Pathogen cross-transmission via building sanitary plumbing systems in a full scale pilot test-rig. PloS one, vol 12, issue 2 (2017) p. e0171556.
[11] Lo, D. C., Liou, J. S., & Chang, S. W. Hydrodynamic performances of air-water flows in gullies with and without swirl generation vanes for drainage systems of buildings. Water, vol 7, issue 2 (2015) pp. 679-696.
[12] Ardejani, F. D., Baafi, E., Panahi, K. S., Singh, R. N., & Shokri, B. J. Application of Computational Fluid Dynamics (CFD) for simulation of acid mine drainage generation and subsequent pollutants transportation through groundwater flow systems and rivers. Computational Fluid Dynamics Technologies and Applications, (2011) pp. 123-160.
[13] Samsuddin, M. A. H. Airflow Analysis of the Bus Under Various Velocities Using CFD Simulation. Advanced and Sustainable Technologies (ASET), vol 3, issue 1 (2024) pp. 80-88.
[14] Tan, K. M. Computational Fluid Dynamics Analysis on the Road Bike Using Different Flow Models under Extreme Inlet Velocity. Advanced and Sustainable Technologies (ASET), vol 3, issue 1 (2024) pp. 62-70.
[15] Paramasivan, S. Analysis of Tesla CyberTruck Speed on the Velocity and Pressure Distribution Using SimFlow Software. Advanced and Sustainable Technologies (ASET), vol 3, issue 1 (2024) pp. 71-79.

Conflict of interest statement: The author declares no conflict of interest.

Author contributions statement: Conceptualization; Methodology; Software; Formal Analysis; Investigation; Writing & Editing, Muhammad Syafiq Bahanuddin.