Explorematics
Journal of Innovative
Engineering and Technology

Volume: 06 No: 01 | April -2025
ISSN (Online) 2636 – 590
ISSN (Print) 2636 - 591X

GEOMETRIC MODELLING AND STRESS ANALYSIS OF FIRE HYDRANT FUNCTIONALITY USING COMPUTER-AIDED DESIGN

Okwuchukwu Innocent Ani*[1], Okwudiri Daniel Ohaa [2], Chinedu Thomas Aniokete[3]
1 Department of Mechanical and Production Engineering, Enugu State University of Science and Technology
2 Enugu State Fire and Rescue Service, Enugu State, Nigeria
3 Department of Chemical Engineering, Faculty of Engineering, Enugu State University of Science and Technology
Author for correspondence: Ani O.I; **E-mail**: okwuchukwu.ani@esut.edu.ng

Abstract - Fire hydrants play a critical role in urban water distribution systems, serving as vital access points for emergency firefighting operations. Their effectiveness hinges on both mechanical reliability and structural resilience. This research project focuses on the geometric modeling and stress analysis of fire hydrants using advanced Computer-Aided Design (CAD) tools to gain a deeper understanding of their functionality and support informed design and maintenance strategies. A comprehensive 3D model of a fire hydrant was developed using AutoCAD, capturing essential features such as the body, valves, flanges, threaded connections, and fittings-all without altering the original design specifications. Realistic materials and textures were applied to enhance visualization and accuracy. The final CAD model was exported in multiple formats to facilitate simulation and presentation. Finite Element Analysis (FEA) was conducted in Abaqus to evaluate the hydrant's structural integrity under simulated operational loads. Results revealed a maximum stress concentration of 5.488 MPa at the junction between the main and branch pipes, with stress decreasing to 0.365 MPa in areas distant from the junction. These stress patterns highlight critical regions influenced by geometric discontinuities and load application. The findings form a basis for design validation, improved maintenance planning, and future structural optimization of fire hydrant systems.
Keywords: Fire hydrant, Computer-Aided Design (CAD), Geometric modelling, Structural analysis

1. Introduction

Fire hydrants play a vital role in urban safety infrastructure, serving as primary access points for emergency firefighting operations. Ensuring their operational reliability, structural resilience, and hydraulic performance is critical, particularly under the high-pressure conditions of fire response scenarios. Recent advancements in Computer-Aided Design (CAD) have significantly transformed the way hydrants are designed, evaluated, and improved. CAD-based geometric modeling offers unprecedented opportunities to enhance fire suppression system performance by enabling precision design, virtual testing, and functional optimization without relying on traditional trial-and-error methods.

CAD tools such as AutoCAD, CATIA, and SolidWorks allow designers to create intricate 2D and 3D models that replicate real hydrant configurations, including mechanical components like valve stems, nozzles, and internal casings. These tools offer deep insights into structural interactions and dynamic performance under varying operational conditions (Kayode & Ajisegiri, 2024). In addition to visualization, CAD modeling now supports integrated stress analysis and fluid flow simulations, offering vital data on potential failure points and performance bottlenecks (Uthayakumar et al., 2025). Through such digital modeling, engineers can identify and resolve structural issues early in the design process, reducing the risk of malfunction during critical emergency situations (Shah et al., 2021).

One of the key benefits of CAD lies in its ability to simulate real-world pressure conditions through virtual environments. This shift from manual drafting saves time and cost as well as significantly improves the accuracy of stress and fluid dynamic evaluations. Simulation platforms such as ANSYS and Autodesk CFD work seamlessly with CAD models to explore how materials and geometries respond to various stress factors. Von-Zeska-de-França et al. (2023) used finite element analysis (FEA) to compare the performance of cast iron and ductile iron in hydrant bodies, finding ductile iron more resistant to impact and internal pressure, a finding that directly informs smarter material selection in design.

Hydraulic performance has also seen notable improvement through CAD-driven modeling. Engineers can simulate water flow, visualize turbulence, and fine-tune discharge patterns without manufacturing physical prototypes. Jalui et al. (2025) demonstrated how changes in internal chamber geometry and nozzle orientation led to smoother flow and enhanced discharge efficiency-critical factors during time-sensitive fire events. Furthermore, parametric modeling within CAD platforms empowers designers to modify dimensions such as stem length or nozzle diameter, enabling custom hydrant configurations suited to specific municipal needs (Shafiee & Berglund, 2015). This flexibility also supports regional standardization efforts, which reduce maintenance costs and streamline parts replacement processes (Piedra-Cascón et al., 2025).

Beyond modeling, the integration of CAD with 3D printing technologies introduces a cost-effective path for prototype production. While printing full-scale, pressure-rated hydrants remains a challenge, producing smaller components or functional mock-ups. This has proven effective for rapid testing. Wen et al. (2018) reported successful fabrication of affordable prototypes using CAD-generated models, dramatically accelerating development timelines. Moreover, CAD tools now enable environmentally conscious design by optimizing internal geometries and reducing material waste, while still maintaining structural integrity. Ergonomics is also gaining attention, as shown in Gomułka's (2023) work that used simulation tools to assess how easily firefighters can operate hydrants under pressure, resulting in more user-friendly designs that prioritize safety and efficiency.

Despite these advancements, many existing hydrant systems still rely on outdated designs lacking the precision and adaptability offered by modern CAD methods. Traditional drafting techniques often fall short in simulating operational stresses or long-term durability (Liu, 2024). Although foundational mechanical studies exist, there remains a significant gap in the use of CAD to simulate how internal hydrant components interact under pressure, manage flow, and endure environmental stresses over time (Heintze et al., 2019). Another critical challenge is the lack of standardized hydrant designs across regions, which complicates repair logistics and raises maintenance costs (Hong & Kong, 2021). Moreover, while some research addresses water discharge efficiency, few studies explore how internal geometry, nozzle alignment, or valve placement can be optimized digitally (Chen & Chen, 2023).

This study addresses these gaps by exploring the power of CAD to build accurate geometric models and perform detailed stress and performance analyses. Through integration with simulation tools, the research investigates how fire hydrant components behave under operational conditions, evaluates material effectiveness, and explores ergonomic and hydraulic improvements. The goal is to advance hydrant designs that are not only structurally sound and hydraulically efficient but also standardized, easier to maintain, and better suited to the evolving demands of urban fire safety systems.

2. Methodology

The methodology adopted for this research is a comprehensive approach integrating Computer-Aided Design (CAD) and Finite Element Analysis (FEA) using AutoCAD and Abaqus software to assess the structural and functional behavior of a fire hydrant under operational conditions. The study focuses on evaluating an existing fire hydrant design rather than proposing a new one. Hence, all modeling and simulations were based on pre-established dimensions sourced from actual hydrant specifications, ensuring accuracy and relevance.

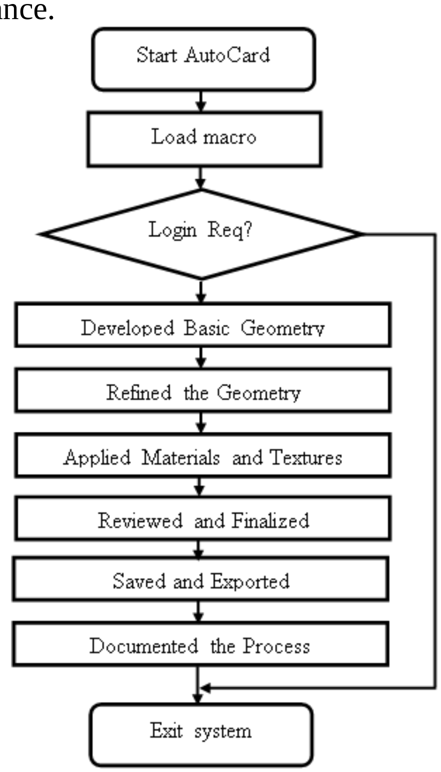

Figure 1: CAD model development flowchart for the fire hydrant system

2.1 Requirement Definition and Conceptual Modeling

The process began with identifying the design and functional specifications of the hydrant. These included external dimensions, connection types, valve placements, and material properties. This information was synthesized into preliminary sketches that laid out the conceptual blueprint of the fire hydrant, highlighting key components such as the main body, flanges, valves, and lugs. The CAD model development flowchart, shown in Figure 1, outlines the full procedural roadmap for building the model.

2.2 3D CAD Modeling with AutoCAD

Using AutoCAD, a new project was initiated with millimeters as the unit of measurement. The coordinate system and grid spacing were carefully adjusted to maintain high accuracy throughout the design process. The modeling started with a cylindrical base representing the main body of the hydrant, followed by modeling the flanges and connection interfaces where the hydrant joins the underground water system.

Advanced extrusion and filleting tools were utilized to shape threads, nozzles, valve ports, and supporting lugs. The dimensioning tools in AutoCAD played a critical role in ensuring all features conformed to real-world tolerances. Thereafter, all components were developed, an assembly model was constructed, accurately placing each part into spatial alignment to reflect actual hydrant operation.

To enhance realism, material textures and surface features were applied to replicate cast iron or ductile iron appearances. This step was essential for both visual verification and rendering purposes. The final CAD model was saved in .dwg format and exported to. stl and .pdf formats for presentation, simulation, and potential 3D prototyping.

2.3 CFD Boundary Definition and Fluid Domain Extraction

After completing the geometric model, the fluid domain within the hydrant was isolated to prepare it for computational simulations. This domain represents the internal pathway that water follows during operation. The inlet and outlet boundary conditions were defined to simulate actual hydrant use which accounts for high-pressure inflow and controlled discharge through the nozzle systems. No geometric modifications were made, preserving the integrity of the original design for an accurate performance analysis.

2.4 Structural and Mechanical Analysis with Abaqus

The CAD model was then imported into Abaqus for Finite Element Analysis (FEA). This stage involved meshing the hydrant geometry and applying boundary conditions to simulate real-world stresses such as internal water pressure, mechanical loads from valve actuation, and environmental forces. Particular attention was given to the junction between the main pipe and branch pipe, which earlier studies indicated as a critical stress concentration point. Results from Abaqus revealed a maximum stress of 5.488 MPa at this junction, while other regions experienced significantly lower stress, around 0.365 MPa, affirming the structural reliability of the design under operational pressures. These insights into stress distribution patterns and deformation responses formed the basis for evaluating the hydrant's performance and guiding future improvements.

2.5 Documentation and Design Validation

The modeling and analysis process was thoroughly documented. This included design decisions, parametric configurations, software settings, and simulation outcomes. By combining CAD-based geometric modeling with advanced FEA techniques, the research provides a validated digital prototype of a fire hydrant, supporting design optimization, maintenance planning, and future upgrades in hydrant development.

3. Results and Discussion

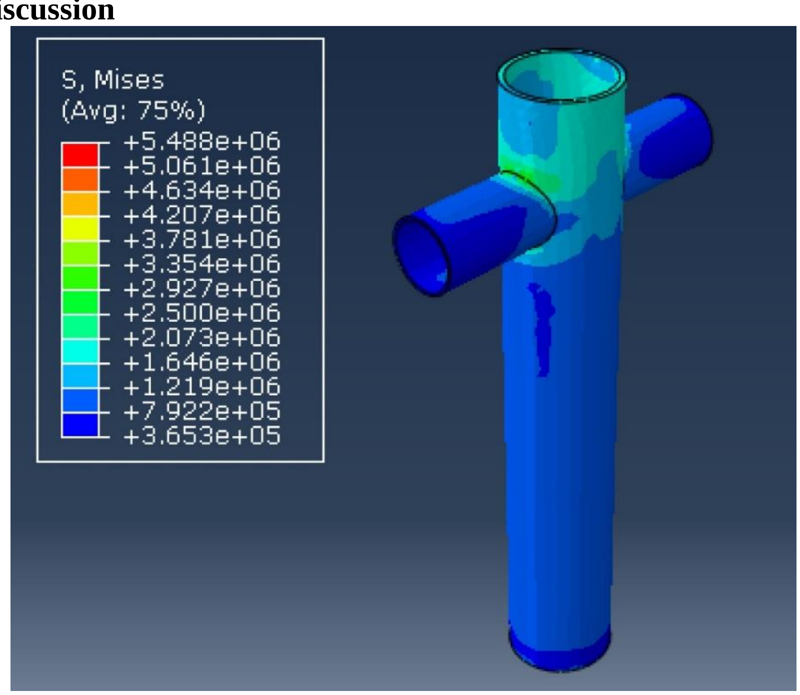

Figure 2: Von Mises stresses

The stress distribution analysis, as illustrated in Figure 2, reveals a maximum Von Mises stress of approximately 5.488 MPa concentrated at the junction between the main pipe and the branch pipes of the fire hydrant structure. Conversely, the minimum stress level of 0.365 MPa is observed in regions away from the junction, particularly along the lower section of the main vertical pipe where the geometry remains uniform and free from complex load transitions. This gradient from high to low stress is indicative of localized stress concentration, a common phenomenon in T-junction geometries where abrupt changes in cross-section lead to discontinuities in load distribution.

The high-stress accumulation at the joint aligns with findings from Gadallah et al. (2020), who reported similar stress localization at weld toe regions in T-joints subjected to axial loads. Their study highlighted that these zones, characterized by sharp geometric transitions and high-stress gradients, are prone to fatigue and potential structural failure over time. This comparison validates the critical nature of junctions in pressure-bearing components like fire hydrants and emphasizes the need for focused reinforcement or design optimization in these areas.

Nevertheless, the research by Ding et al. (2024) introduces a contrasting perspective. Their work demonstrated that the integration of fillet reinforcements in T-joints significantly reduced the stress concentration factor, offering a smoother transition for load transfer and thereby distributing the stress more evenly. Applying this insight to the current hydrant model suggests a potential design improvement-incorporating fillets or curved transitions at junction interfaces may help mitigate peak stresses and enhance the hydrant's durability under cyclic loading or emergency pressure surges.

The use of Abaqus simulation software enabled precise identification of these critical stress regions, affirming the value of digital analysis in early-stage product evaluation. The current model's ability to highlight failure-prone zones without physical prototyping demonstrates an efficient, cost-effective strategy for enhancing fire hydrant performance and guiding future iterations. This study shows the confirmation that geometric discontinuities, especially at intersection points, are major contributors to structural stress**.** Design revisions focusing on smoother transitions or the application of structural reinforcements could significantly reduce localized stress peaks to improve the safety and longevity of the hydrant system.

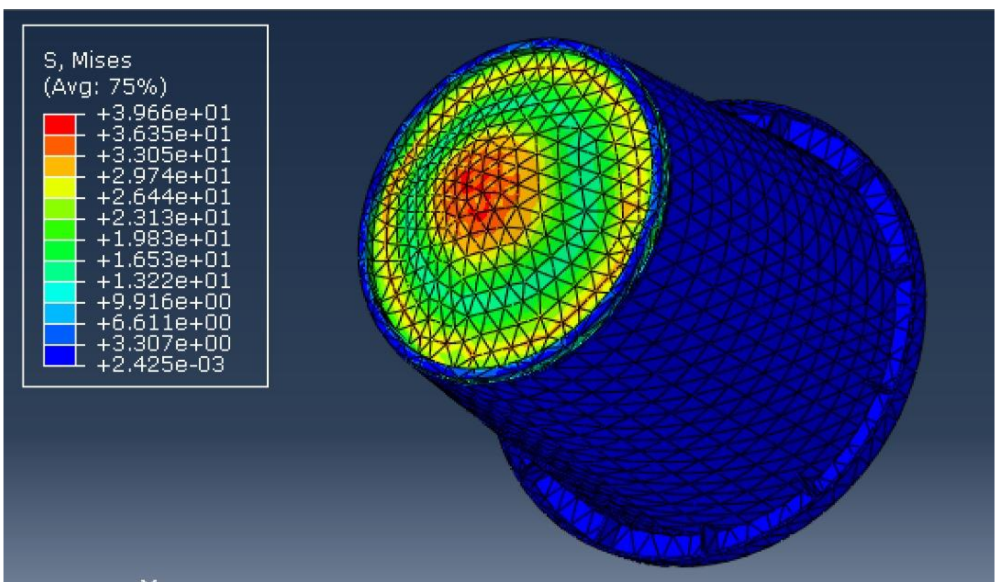

Figure 3: Von Mises stresses at the cap of the fire hydrant.

The finite element analysis (FEA) results, as depicted in Figure 3, present the Von Mises stress distribution at the fire hydrant's outlet, capturing essential mechanical behavior under operational load conditions. The observed stress values range from 2.425×10^{-3} MPa to 39.66 MPa, with stress peaking prominently at the central outlet surface. This region is subject to direct fluid pressure and structural constraints, making it highly susceptible to stress convergence and concentration. The central stress concentration observed is in agreement with findings from Caliendo and Russo (2022), who emphasized that connection points in fluid-bearing outlets often experience heightened stress levels due to the convergence of dynamic pressure and structural resistance. This correlation reinforces the current model's indication that design geometry around fluid exits requires particular attention to avoid premature structural failures.

In contrast, Lu et al. (2018) reported that maximum stress tends to accumulate around flange regions, particularly due to suboptimal bolt configurations. Their study highlighted that bolt misalignment and uneven distribution shift load paths away from the outlet center. The current simulation shows minimal stress intensity at the flange and edge zones, suggesting adequate load distribution in the modeled design and possibly superior flange integrity compared to previously analyzed systems.

Furthermore, the stress range between 29.74 MPa and 39.66 MPa matches closely with results by Venkatesan et al. (2016), who explored outlet stress responses under cyclic loading conditions. Their findings confirmed that core outlet regions consistently exhibit higher mechanical vulnerability, especially when subjected to recurring pressure surges. This is a condition characteristic of firefighting scenarios and this resemblance in stress profile solidifies the credibility of the present FEA setup as well as confirming the criticality of the outlet zone in hydrant durability assessments.

However, not all studies agree. Osaragi and Hirokawa (2019) observed relatively uniform stress distributions across hydrant structures, attributing this to the use of anisotropic materials that spread stress more evenly. The present results diverge from this, likely due to the isotropic assumption in material properties during modeling and the presence of geometric discontinuities that concentrate load paths.

The relatively lower stress levels around the perimeter, particularly those below 9.916 MPa, reinforce the idea that structural geometry strongly influences stress flow paths. Rounded transitions and material thickness play vital roles in dispersing pressure loads, minimizing the formation of sharp gradients in non-critical areas.

These insights underscore the value of FEA simulations in identifying mechanical weak points. By accurately pinpointing areas of peak stress, such as the hydrant outlet, designers can develop targeted reinforcement strategies, optimize geometries, or integrate stress-relieving features such as fillets or tapering. It is important to ensure mechanical integrity at these zones for the long-term reliability and safety of hydrant systems. This study demonstrates the FEA simulation of the fire hydrant outlet. It clearly identifies the central outlet region as a critical stress zone, validating existing literature on pressure-driven mechanical behavior. The insights gained support strategic reinforcement and provide a reliable foundation for optimizing hydrant design to meet operational and safety standards under high-pressure firefighting situations.

4. Conclusion

This research has successfully demonstrated the transformative potential of Computer-Aided Design (CAD) and finite element analysis (FEA) in evaluating and optimizing fire hydrant systems. Through the integration of CAD software for geometric modeling and Abaqus for stress analysis, the project provided a detailed and accurate recreation of a functional fire hydrant, capturing essential engineering components such as threads, flanges, valves, and connection points. These digitally modeled elements not only align with existing hydraulic specifications but also serve as the foundation for advanced performance assessments under operational loads.

The structural analysis revealed critical stress concentration zones, particularly at the junctions and outlet centers, validating previous findings in literature and emphasizing the need for design reinforcement in high-stress areas. By avoiding the need for physical prototypes in early design stages, the CAD and FEA-based methodology greatly reduces development costs and time, while improving the accuracy of performance predictions. Furthermore, the digital workflow supports rapid iteration**,** material testing, and customization to meet specific site or environmental demands with capabilities that traditional methods fail to deliver efficiently.

Importantly, the study underlines how CAD-driven modeling not only enhances fire hydrant reliability but also opens the door for similar innovations in other utility infrastructures**.** Systems such as water valves, pressure regulators, or underground fittings can benefit from this digital approach, leading to smarter urban planning, reduced maintenance costs, and improved public safety infrastructure. The combination of virtual testing, real-time stress visualization, and parametric design flexibility proves invaluable in creating next-generation hydrant systems that are safer, more efficient, and better suited for modern firefighting demands**.**

In conclusion, this research establishes a robust foundation for digital transformation in mechanical utility design**,** advocating the adoption of CAD and simulation tools as standard practice for engineers aiming to balance cost, performance, and innovation in public infrastructure development.

References

Caliendo, C., & Russo, I. (2022). A 3D computational fluid dynamics model for assessing the concrete spalling of a tunnel lining in the event of a fire. Computers and Geotechnics, 152, 105041.

Ding, H., Zhang, W., Zhang, Z., Yin, D., He, W., & Xie, D. (2024). Influence of welding sequences and boundary conditions on residual stress and residual deformation in DH36 steel T-joint fillet welds. Thin-Walled Structures, 204, 112337.

Gadallah, R., Tsutsumi, S., Tanaka, S., & Osawa, N. (2020). Accurate evaluation of fracture parameters for a surface-cracked tubular T-joint taking welding residual stress into account. Marine Structures, 71, 102733.

Gomułka, P. (2023). Computer-aided system for layout of fire hydrants on boards designed vessel using the Particle Swarm Optimization algorithm. Polish Maritime Research, 30(4), 4-16.

Heintze, S. D., Reichl, F. X., & Hickel, R. (2019). Wear of dental materials: Clinical significance and laboratory wear simulation methods-A review. Dental materials journal, 38(3), 343-353.

Hong, S. H., & Kong, H. S. (2021). Analysis of the Number of Kinks with the Fire Hose Staking Method of Indoor Hydrant System. Journal of the Korea Safety Management & Science, 23(4), 61-66.

Jalui, S., Xuan, Y., Manogharan, G., & O'Connor, J. (2025). Development of a Method for Shape Optimization for a Gas Turbine Fuel Injector Design Using Metal-Additive Manufacturing. Journal of Engineering for Gas Turbines and Power, 147(1).

Kayode, A. J. B., & Ajisegiri, E. (2024, April). Engineering Transformation: From Computer Aided Design to the Cloud. In 2024 International Conference on Science, Engineering and Business for Driving Sustainable Development Goals (SEB4SDG) (pp. 1-16). IEEE.

Liu, Z. (2024). Comparison and analysis of advantages and disadvantages between BIM and CAD in civil drafting software. Applied and Computational Engineering, 62, 192-197.

Lu, H., Ma, G., Li, X., & Wu, S. (2018). Stress analysis of LNG storage tank outlet pipes and flanges. Energies, 11(4), 877.

Osaragi, T., & Hirokawa, N. (2019). Simulation Analysis of Fire Hydrant Usability Levels after Large Earthquake. In ISCRAM.

Piedra-Cascón, W., Burgos-Artizzu, X. P., González-Martin, Ó., Oteo-Morilla, C., Pose-Rodriguez, J. M., & Gallas-Torreira, M. (2025). Evaluation of the accuracy (trueness, precision) and processing time of different 3-dimensional CAD software programs and algorithms for virtual cast alignment. Journal of Dentistry, 155, 105619.

Shafiee, M. E., & Berglund, E. Z. (2015). Real-time guidance for hydrant flushing using sensor-hydrant decision trees. Journal of Water Resources Planning and Management, 141(6), 04014079.

Shah, G. A., Polette, A., Pernot, J. P., Giannini, F., & Monti, M. (2021). Simulated annealing-based fitting of CAD models to point clouds of mechanical parts' assemblies. Engineering with Computers, 37(4), 2891-2909.

Uthayakumar, G. S., Yaramadhi, M., Marimuthu, T., & Lakshmi, T. V. (2025). Optimized Mixed-Order Relation-Aware Recurrent Neural Networks based CAD Model for Skin Cancer Detection and Classification. Knowledge-Based Systems, 315, 113222.

Venkatesan, B., Ilangovan, R., Jayabalan, P., Mahendran, N., & Sakthieswaran, N. (2016). Finite element analysis (FEA) for the beam-column joint subjected to cyclic loading was performed using ANSYS. Circuits and Systems, 7(08), 1581.

Von-Zeska de França, G., de Assumpção, R. L., Luersen, M. A., & da Silva, C. H. (2023). A Bidimensional Finite Element Study of Crack Propagation in Austempered Ductile Iron. Computers, Materials & Continua, 77(2).

Wen, X., Gao, S., Feng, J., Li, S., Gao, R., & Zhang, G. (2018). Chest-wall reconstruction with a customized titanium-alloy prosthesis fabricated by 3D printing and rapid prototyping. Journal of cardiothoracic surgery, 13, 1-7.