Indian Journal of Production and Thermal Engineering (IJPTE)
ISSN: 2582-8029 (Online), Volume-3 Issue-6, October 2023

Airflow Simulation in a Turbofan Engine: A Study of Flow Behavior

Ahmad Farzad Faqiri, Noman Tasal, Aminullah Bakhtyar, Mahboobullah Mutahar, Suhrab Sheybani

Abstract*: The efficient functioning of modern turbofan engines relies heavily on a deep understanding of airflow dynamics within critical components. This research paper presents a comprehensive investigation into airflow simulation within a turbofan engine, employing advanced computational techniques. The study focuses on flow behavior and makes use of SolidWorks for 3D modeling and ANSYS for simulation. The investigation centers on analyzing key flow parameters such as velocity, pressure, and temperature. The methodology involves creating an accurate 3D model of the turbofan engine excluding the compressor, combustion chamber, and turbine using SolidWorks to capture fine geometry and details. Subsequently, ANSYS is utilized to simulate the airflow within the turbofan engine, simulating realistic conditions and enabling the detailed analysis of flow behavior. The results of this study advance knowledge of turbofan engine technology and lay the groundwork for additional study and advancement in the area of aviation propulsion systems. In response to the evolving requirements of the aviation sector, the knowledge acquired from this research will serve as a priceless asset in the development of engines that are characterized by enhanced dependability, a reduced ecological footprint, and heightened fuel efficiency.

***Keywords*: Airflow Simulation, Turbofan Engine, Computational Fluid Dynamics (CFD), SolidWorks, ANSYS.**

I. INTRODUCTION

The aviation industry has witnessed remarkable advancements in propulsion systems, with turbofan engines being the driving force behind modern aircraft performance [1]. As air travel continues to grow, there is an increasing demand for more efficient and reliable engines that can meet stringent environmental regulations while ensuring enhanced operational capabilities [2] [3]. In this context, the study of airflow behavior in turbofan engine assumes utmost significance as it directly impacts engine performance and efficiency. Computational fluid dynamics (CFD) simulations have emerged as indispensable tools for understanding and optimizing the intricate flow patterns within such engines [4].

This study delves into an in-depth exploration of airflow simulation within a turbofan engine, with the goal of examining flow patterns. The process involves employing SolidWorks to create a 3D model of the turbofan engine. Furthermore, the analysis of these simulations makes use of ANSYS, a widely recognized platform for computational fluid dynamics (CFD). This combination facilitates a meticulous investigation into the intricate flow dynamics within the engine's confines. Over the past decade, significant advancements on airflow simulations in turbofan engine have provided valuable insights into flow dynamics and their effects on engine performance. Numerous research papers have examined various aspects of engine flow behavior. Here are a few researchers who are involved in turbofan engine airflow simulations: Hassan et al. investigates the impact of fan efficiency on modern high bypass turbofan engines, focusing on the CF6-50 engine's transonic axial flow fan and intake under takeoff conditions. Using accurate dimensions and computational fluid dynamics, shock waves are observed forming at the suction side's trailing edge, propagating towards the opposite blade's pressure side [5]. Roberts et al. developed a new transient two-stream engine model by combining individual component models in MATLAB and Simulink. The model includes the fan, high-pressure compressor, combustor, high-pressure turbine, low-pressure turbine, plenum volumes, and exit nozzle. The focus was on enhancing transient capabilities, improving physics models, eliminating algebraic constraints, and reducing simulation time using advanced numerical solvers [6] [7]. Garcia Rosa et al. study the air flow through the fan stage of a high-bypass, geared turbofan during windmilling conditions, which is an off-design operation. The experiments were conducted in a turbofan test rig of ISAE, specifically designed to simulate windmilling operation in a ground setup. The study also explores the relationship between fan rotational speed and mass flow rate, finding that they are proportional. However, the fan rotor's inlet and outlet relative flow angles and the fan load profile remain constant for different mass flow rates. The impact of flow behavior on engine efficiency and emissions has been a prominent area of interest [8] [9]. Mark et al. utilized ANSYS 14.5 software to conduct numerical simulations, enabling them to investigate aerodynamic flow characteristics, air-fuel mixture, combustion turbulence, thermal, and cooling analysis [10].

Manuscript received on 26 August 2023 | Revised Manuscript received on 08 September 2023 | Manuscript Accepted on 15 October 2023 | Manuscript published on 30 October 2023.
***Correspondence Author(s)**
Aminullah Bakhtyar*, Department of Mechanical Engineering, Kabul University, Kabul, Afghanistan. Email: bakhyar.aminullah97@gmail.com, ORCID ID: 0009-0005-1187-818X.
Ahmad Farzad Faqiri, Department of Mechanical Engineering, Kabul University, Kabul, Afghanistan. Email: farzad.faqiri00@gmail.com, ORCID ID: 0009-0001-3782-7664.
Noman Tasal, Department of Mechanical Engineering, Kabul University, Kabul, Afghanistan. Email: noman.tasal00@gmail.com, ORCID ID: 0009-0007-5066-4912.
Mahboobullah Mutahar, Department of Mechanical Engineering, Kabul University, Kabul, Afghanistan. Email: afmutahar@gmail.com, ORCID ID: 0009-0000-0917-5091.
Suhrab Sheybani, Department of Mechanical Engineering, Kabul University, Kabul, Afghanistan. Email: suhrabsheybani10@gmail.com, ORCID ID: 0009-0007-8339-5827.

Guimarães et al. examine the behavior of a complex distorted inflow in a full-scale engine rig using a 21-inch diameter StreamVane to generate prescribed secondary flow distributions. They employ a vortex dynamics-based model called StreamFlow to predict the mean secondary flow development based on experimental data [11]. Qian et al. developed a mathematical model for a turbofan engine and conducted simulation calculations. They created an engine component model based on the gas flow path and established common working equations for typical conditions. To solve the nonlinear equations, they employed the N+1 residual method. Using design point parameters, the authors formulated a general engine model and calculated specific performance parameters under standard atmospheric conditions [12]. Yang et al. created a platform for evaluating the secondary air system's (SAS) performance, focusing on internal flow and heat transfer analysis. The platform employs a multi-fidelity modeling technique for a turbofan engine under various flight conditions. It introduces a turbine blade cooling model that combines external heat transfer calculations with coolant side modeling. Computational Fluid Dynamics (CFD) is used to capture the complex flow field structure in the preswirl system [13]. These studies build upon their findings to offer a study of flow behavior under operating conditions for turbofan engines. In a time characterized by growing requests for air travel that is both environmentally friendly and optimized for performance, possessing a profound comprehension of the fluid dynamics inside a turbofan engine emerges as exceptionally crucial. By scrutinizing key flow parameters, including velocity, pressure, and temperature, under realistic operating conditions, this research contributes to the broader goal of optimizing engine design and performance.

The subsequent sections of this paper will detail the methodology adopted for 3D modeling and simulation, explain the choice of SolidWorks and ANSYS as tools, and present a systematic approach for analyzing the obtained results. Ultimately, this investigation aims to provide a comprehensive framework for studying airflow phenomena within the turbofan engine, offering insights that have the potential to reshape the future of aviation propulsion systems.

II. METHODOLOGY

The methodology entails creating a comprehensive three-dimensional representation of a turbofan engine through SolidWorks 2023 software. This model deliberately omits certain elements like compressors, combustion chambers, turbines, and stator vanes. ANSYS 2023 R1 is then leveraged to conduct in-depth numerical analyses, focusing on velocity, pressure, and temperature distributions. The simulation setup involves defining boundary conditions, meshing the model, and applying relevant physics. Advanced turbulence models and solver configurations are selected to accurately capture airflow dynamics. The simulation results are meticulously post-processed and analyzed, providing insights into the aerodynamic behavior of the engine. This combined methodology not only ensures a comprehensive investigation of the airflow within the turbofan engine but also demonstrates a robust framework for studying complex aerospace systems, facilitating future advancements in aviation technology. The subsequent sections will present the detailed outcomes of this methodology, shedding light on the intricate aerodynamics governing the engine's operation.

A. Geometric Modeling of Turbofan Engine

The initial phase involves using SolidWorks' powerful and intuitive design capabilities to craft a detailed three-dimensional geometric model of the turbofan engine. Compressors, combustion chambers, turbines, and stator vanes are not included in the model since they complicate modeling. The geometry accurately represents the fan, shaft, casing, nacelle, and other relevant features. The geometric model is rigorously validated against existing engine specifications to guarantee accuracy and reliability in performance. Fig. 1 shows the final Solid Works model turbofan engine.

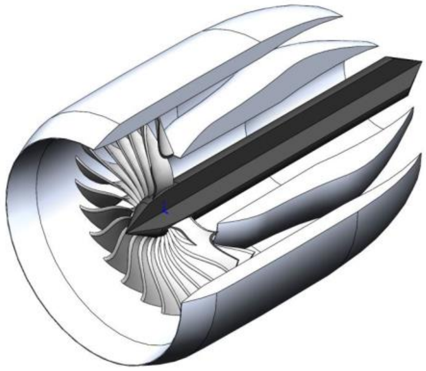

Fig. 1: SolidWorks Model of Turbofan Engine

B. Simulation Setup

After importing geometry into ANSYS, an essential addition to the methodology is the creation of a rotating region meticulously defined around the fan, capturing its rotational dynamics. This enables a realistic representation of the fan's interaction with the surrounding airflow. Additionally, a computational domain is established around the turbofan engine, encompassing the pertinent flow regions and facilitating a comprehensive analysis. Fig. 2 represents the view of the turbofan engine in ANSYS.

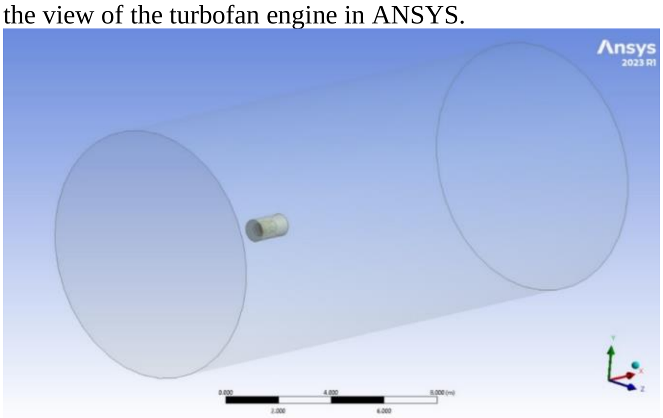

Fig. 2: ANSYS View

ANSYS Meshing is harnessed for the generation of an unstructured computational mesh enveloping the geometry. This unstructured meshing approach confers flexibility in resolving intricate flow regions while preserving computational efficiency. The mesh undergoes a meticulous series of quality assessments and refinement stages to ensure the production of dependable simulation outcomes. Fig. 3 presents the mesh layout.

Retrieval Number: 100.1/ijpte.C79050912323
DOI: 10.54105/ijpte.C7905.103623
Journal Website: www.ijpte.latticescipub.com

Published By:
Lattice Science Publication (LSP)

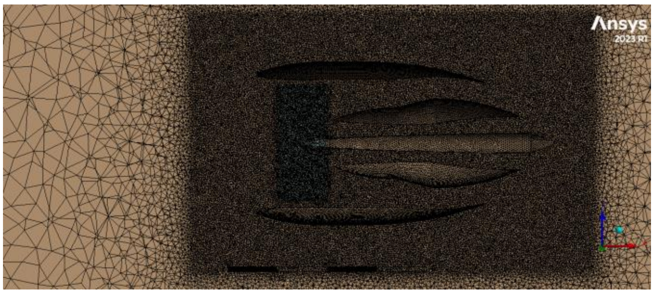

Fig. 3: Mesh Layout

The boundary conditions for the simulation are defined based on the engine's operating conditions. The computational domain contains a velocity inlet, pressure outlet, wall for the intake wall, and side for the far field, and these are specified to replicate realistic operational scenarios. Also, the MRF method is used to define turbofan rotation. Appropriate turbulence models, such as k-omega, are selected to capture turbulent flow behavior. Table- I lists the boundary conditions.

Table- I: Boundary Conditions

Fan speed	0.8 Mach
Temperature	294 K
Velocity inlet	5 m/s
Pressure outlet	0 Pa
Air density	1.145 kg/m³

ANSYS Fluent is employed as the computational fluid dynamics (CFD) solver for simulating airflow within the turbofan engine. The governing Navier-Stokes equations are solved iteratively using implicit or explicit schemes, depending on stability and convergence considerations. Solver settings, convergence criteria, and time-stepping methods are fine-tuned to ensure accurate and efficient simulations.

III. RESULTS

Once the simulations are completed, extensive post-processing is conducted to extract relevant data and visualize flow phenomena. Velocity, pressure distribution, and temperature profiles are among the key parameters analyzed and the results are provided in below.

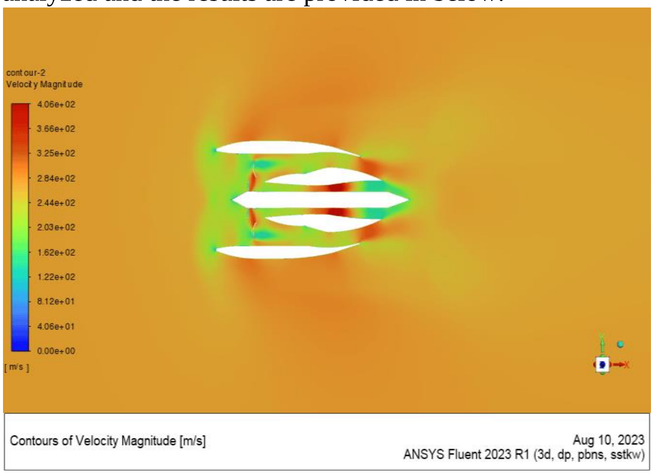

Fig. 4: Velocity Magnitude Contour

The velocity contour depicted in Fig. 4 offers a comprehensive insight into the intricate flow behavior observed throughout the turbofan engine. As the air traversed the engine components, a complex pattern of acceleration and deceleration was evident. Upon passing through the fan, the air experienced an initial acceleration, with velocity increasing progressively. This accelerated flow encountered further acceleration in the bypass section, culminating in the highest velocity attained at the outlet. The analogy of a track race aptly characterizes this phenomenon, where air particles compete for swiftness. Intriguingly, the core flow within the engine exhibited a distinct acceleration, reaching its peak velocity within the confines of the combustion chamber. Subsequent to this combustion-driven acceleration, the flow decelerated upon entering the turbine. Notably, the two distinct flows amalgamated before exiting the engine, resulting in an egress velocity nearly mirroring the initial inlet speed.

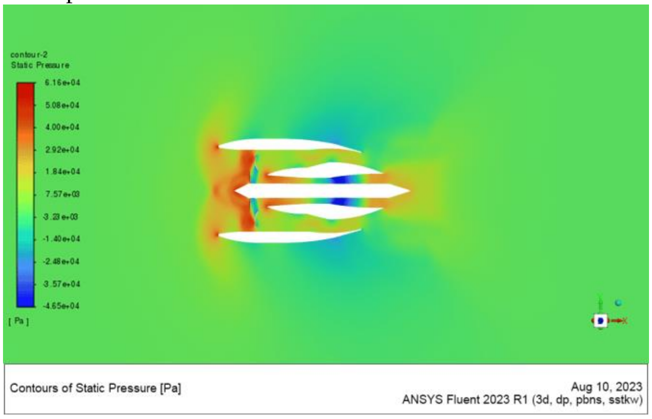

Fig. 5: Static Pressure Contour

Complementing the velocity insights, the pressure contour depicted in Fig. 5 adds a layer of understanding to the behavior of the airflow. The areas of heightened pressure, notably preceding the fan and along the bypass channels, offer vital information about the aerodynamic dynamics. These pressure gradients contribute to flow redirection and interaction within the engine, ultimately influencing performance and efficiency.

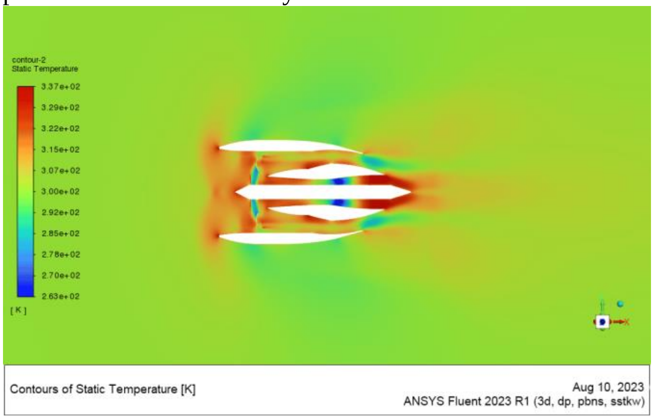

Fig. 6: Static Temperature Contour

The temperature contour provided in Fig. 6 contributes to the multifaceted analysis of the engine's behavior. A direct relationship between temperature and pressure is evident, as the temperature rises in correspondence with the higher-pressure regions identified in the pressure contour.

Retrieval Number: 100.1/ijpte.C79050912323
DOI: 10.54105/ijpte.C7905.103623
Journal Website: www.ijpte.latticescipub.com

Published By:
Lattice Science Publication (LSP)

The heating process, manifesting as an increase in temperature as the air progresses through the engine, is a result of the compression and combustion processes. This interplay of temperature and pressure provides insights into the thermodynamic intricacies of the engine's operation.

IV. DISCUSSION

When considered collectively, previous contours present a holistic view of airflow within the turbofan engine. The acceleration, deceleration, pressure variations, and temperature changes collectively illustrate the complex interplay between aerodynamics and thermodynamics, contributing to the engine's overall performance. This research, conducted through the collaborative employment of SolidWorks and ANSYS, unveils valuable insights into the behavior of air within the turbofan engine. The implications of these findings extend beyond the realm of theoretical understanding; they hold the potential to influence real-world engineering endeavors. Engine performance optimization, design enhancement, and aircraft efficiency are just a few of the avenues where these insights could make a tangible impact.

Anticipating the road ahead, upcoming inquiries may plunge further into the less-explored facets of the turbofan engine, encompassing the compressor, combustion chamber, and turbine elements. Developing a thorough grasp of these constituents, along with the knowledge acquired from this current examination, holds the potential for a more exhaustive appreciation of the engine's operations. As the voyage of exploration endures, the harmonious culmination of discernments from both hands-on experimentation and computerized analyses will unquestionably foster progress in the domain of aviation and propulsion innovations.

V. CONCLUSION

This research article delved into the analysis of flow patterns within a turbofan engine. The investigation specifically honed in on parameters like velocity, pressure, and temperature while the engine was operating. The discoveries made via this inquiry will have a significant impact on the development of aviation propulsion systems. Improved efficiency in conjunction with enhanced fuel economy and emissions controls pave the way for the development of novel aircraft propulsion systems. These innovations enable the creation and fine-tuning of engines that offer heightened reliability, contributing to air travel that is both trustworthy and gentle on the environment. The SolidWorks model's simplicity and the simulation technique' built-in presumptions are limitations. For deeper insights, future study may employ more sophisticated models. Understanding engine flow behavior requires a critical step, which the focus on airflow simulation in turbofan engines represents. SolidWorks and ANSYS proved a powerful combination in unraveling engine performance complexities, supporting the aviation industry in achieving efficient and eco-friendly aircraft engines, and ensuring safer and more sustainable air travel for future generations.

DECLARATION STATEMENT

Funding	No, we did not receive
Conflicts of Interest	No conflicts of interest to the best of our knowledge.
Ethical Approval and Consent to Participate	No, the article does not require ethical approval and consent to participate with evidence.
Availability of Data and Material	Not relevant.
Authors Contributions	All authors having equal contribution for this article.

REFERENCES

1. Liu, X. et al. "Development and progress in aeroacoustic noise reduction on turbofan aeroengines." *Progress in Aerospace Sciences* 130, 100796 (2022). https://doi.org/10.1016/j.paerosci.2021.100796
2. Guimarães, T., Todd Lowe, K. & O'Brien, W. F. "Complex Flow Generation and Development in a Full-Scale Turbofan Inlet." *J Eng Gas Turbine Power* 140, (2018). https://doi.org/10.1115/1.4039179
3. Gijare, O., Khaire, S., Bagwan, Mohd. Y., Srivastava, S., & Kulkarni, Prof. V. Productivity Optimization in a Shopping Complex using Industrial Engineering and Facility Layout Planning Tools. In International Journal of Management and Humanities, Vol. 4, Issue 8, pp. 35–39 (2020). https://doi.org/10.35940/ijmh.h0766.044820
4. Laskowski, G. M. et al. "Future Directions of High Fidelity CFD for Aerothermal Turbomachinery Analysis and Design." *in 46th AIAA Fluid Dynamics Conference (American Institute of Aeronautics and Astronautics, 2016)*. https://doi.org/10.2514/6.2016-3322
5. H. Z. Hassan, M. H. Gobran & A. Abd El-Azim. "3-D Flow Simulation through the Intake and Fan of Turbofan Engine at Take-Off Conditions." *International Review of Aerospace Engineering* 6, 1–8 (2013).
6. Roberts, R. A. & Eastbourn, S. M. "Modeling Techniques for a Computational Efficient Dynamic Turbofan Engine Model." *International Journal of Aerospace Engineering* 2014, 1–11 (2014). https://doi.org/10.1155/2014/283479
7. Rakesh, N. L., R, S., & Karthikeyan, S., Performance Analysis of Jet Engine of Aircraft. In International Journal of Engineering and Advanced Technology, Vol. 8, Issue 6s2, pp. 606–610 (2019). https://doi.org/10.35940/ijeat.f1179.0886s219
8. García Rosa, N., Dufour, G., Barènes, R. & Lavergne, G. "Experimental Analysis of the Global Performance and the Flow Through a High-Bypass Turbofan in Windmilling Conditions." *J Turbomach* 137, (2015). https://doi.org/10.1115/1.4028647
9. Alam*, I. F., & Legowo, N., ERP Performance Measurement AT PT Hero Supermarket using the IT Balanced Scorecard Method. In International Journal of Recent Technology and Engineering (IJRTE), Vol. 8, Issue 4, pp. 8375–8379 (2019). https://doi.org/10.35940/ijrte.d9062.118419
10. Mark, C. P. & Selwyn, A. "Design and analysis of annular combustion chamber of a low bypass turbofan engine in a jet trainer aircraft." *Propulsion and Power Research* 5, 97–107 (2016). https://doi.org/10.1016/j.jppr.2016.04.001
11. Qian, R., Li, B., Song, H., Dong, Q. & Wang, Y. "Aerodynamic Thermodynamic Modeling and Simulation of Turbofan Engine." *IOP Conf Ser Mater Sci Eng* 685, 012024 (2019). https://doi.org/10.1088/1757-899X/685/1/012024
12. YANG, X., JIAN, M., DONG, W. & XU, Q. "Simulation of the secondary air system of turbofan engines: Insights from 1D-3D modeling." *Chinese Journal of Aeronautics* 36, 231–245 (2023). https://doi.org/10.1016/j.cja.2022.06.018
13. Murtaza*, Dr. M. A., Aerodynamic Performance of Road Vehicles. In International Journal of Innovative Technology and Exploring Engineering, Vol. 9, Issue 6, pp. 642–645 (2020). https://doi.org/10.35940/ijitee.f3327.049620

AUTHORS PROFILE

Ahmad Farzad Faqiri is a dedicated mechanical engineer and researcher holding a B.Sc. degree in Mechanical Engineering from Kabul University. He recently published a review article, demonstrating his analytical prowess and ability to synthesize research. As a student member of the American Society of Mechanical Engineers (ASME), Ahmad Farzad remains actively engaged in the mechanical engineering community, staying updated on the latest developments and networking with professionals. His achievements include both his publication and ASME membership, underscoring his commitment to innovation and progress in the field.

Noman Tasal, a graduate of Mechanical Engineering from Kabul University, Afghanistan, is a visionary engineer whose research spans diverse fields. During his university years, he led successful projects on obstacle-detecting vacuum cleaners and innovative traction machines for lumber disc treatment. His impactful publications delve into robotics and biomechanics. Tasal envisions a future of seamless human-technology integration, especially in wearable robotics. His journey showcases a commitment to innovation that continues to drive meaningful advancements in engineering.

Aminullah Bakhtyar is a distinguished mechanical engineer who graduated from Kabul University, Afghanistan. During his studies, he exhibited a remarkable aptitude for research and innovation, resulting in the creation of innovative machines like an obstacle-detecting automatic vacuum cleaner and a progressive treatment device for lumbar disc ailments. Aminullah's academic journey culminated in the publication of a groundbreaking research paper on assistive wearable lower limb robotics in the International Journal of Latest Engineering and Management Research (IJLEMR) in July 2023. Actively engaged in engineering societies, Aminullah's legacy is one of relentless dedication, creativity, and a commitment to advancing engineering solutions for real-world challenges.

Mahboobullah Mutahar is a dedicated Senior Mechanical Engineering student at Kabul University's Faculty of Engineering in Afghanistan, showcasing exceptional aptitude for innovation and robotics. His remarkable journey includes the publication of "Assistive Wearable Lower Limb Robot," a testament to his profound grasp of robotics and biomechanics. Beyond academics, Mahboobullah actively engages in interdisciplinary projects, demonstrating strong teamwork and practical problem-solving. His memberships in engineering and robotics clubs underline his commitment to holistic learning, while his leadership in workshops and seminars reflects his dedication to community engagement. With analytical acumen and creative zeal, Mahboobullah is not only a standout student but also a promising contributor to the advancement of assistive technologies in mechanical engineering.

Suhrab Shyebani holds a B.Sc. degree in Mechanical Engineering from Kabul University and currently serves as a skilled Plumber Engineer at Takhar Municipality. With a focus on innovative plumbing solutions and sustainable water management, his research contributions have been featured in reputable journals and conferences. His achievements include implementing water-efficient plumbing systems in municipal projects and developing sustainable plumbing designs for community centers. Shyebani's commitment to engineering excellence underscores his dedication to advancing plumbing practices and infrastructure development for the betterment of communities.