International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347-6435 (Online) Volume 15, Issue 03, March 2026)

Numerical Simulation of Porous Injectors in Liquid Rocket Engines

Abhijith[1], Harika[2], Venkatesh[3], Prof. V Phaninder Reddy[4]
[1,2,3]*Student, Dept. of Aeronautical Engineering, IARE, Hyderabad, India*
[4]*Guide, Dept. of Aeronautical Engineering, IARE, Hyderabad, India*

***Abstract*— Porous injectors in liquid rocket engines is a critical area of research aimed at enhancing the performance and efficiency of propulsion systems. Porous injectors are utilized to achieve fine atomization and uniform distribution of propellants, which are essential for stable combustion and optimal thrust generation. This study focuses on the development and application of advanced computational fluid dynamics (cfd) techniques to model the complex multiphase flow dynamics within porous injectors. The simulations account for the intricate interactions between the liquid propellant and the porous medium, including capillary forces, surface tension, and turbulent mixing. This research underscores the importance of numerical simulation as a tool for advancing the understanding and development of next-generation propulsion technologies.**

***Keywords*— Porous Injectors, Atomization, Liquid Rocket Engines, computational fluid dynamics (cfd).**

I. INTRODUCTION

1.1 Background

Porous injectors represent an alternative injection concept to coaxial injectors for rocket engine applications using gas/liquid propellant combinations such as liquid oxygen (LOX)/hydrogen (H_2). This paper summarizes the main design features of porous injectors and proposes a mechanism of atomization for porous injectors that is considerably different to the atomization mechanism for coaxial injectors. The results of hot-fire test campaigns are presented, in which several parameters relevant to the injection process were varied :the injection velocities and momentum fluxes, the combustion chamber Machnumberat the beginning of the nozzle contraction, and the LOX inject or diameter. All hot-fire tests were conducted at P8 test facility for high pressure combustion research at the DLR site of Lampoldshausen with 50-mm-diameter combustion chambers operated with LOX/H_2 at sub- and supercritical chamber pressures between 30 and 100 bar. The results presented here are supporting the proposed mechanism of atomization and allow derivation of some general design guidelines for porous injectors.

This study focuses on the numerical simulation of a porous nozzle using computational fluid dynamics (CFD) techniques. Simulations are performed to analyze the flow characteristics. Residual pressure and fuel distribution within a porous structure. Multiple levels of porosity Nozzle shape and operating conditions are modeled to understand their influence on flow dynamics and mixing performance.

1.2 Theory

1.2.1 Boundary Conditions

Boundary conditions are a crucial aspect of numerical simulations, as they define the constraints and interactions between the simulation domain and the external environment. Here's a detailed explanation of the boundary conditions for the numerical simulation of porous injectors in liquid rocket engines:

Inlet Boundary Conditions:

1. *Pressure Inlet:* The pressure at the inlet is typically specified as a constant value or a time-dependent function. This boundary condition is used to simulate the fuel and oxidizer supply pressures.

2. *Velocity Inlet:* The velocity at the inlet is specified as a constant value or a time-dependent function. This boundary condition is used to simulate the fuel and oxidizer flow rates.

3. *Temperature Inlet:* The temperature at the inlet is specified as a constant value or a time-dependent function. This boundary condition is used to simulate the fuel and oxidizer temperatures.

4. *Species Concentration Inlet:* The species concentration at the inlet is specified as a constant value or a time-dependent function. This boundary condition is used to simulate the fuel and oxidizer composition.

Outlet Boundary Conditions:

1. *Pressure Outlet:* The pressure at the outlet is typically specified as a constant value or a time-dependent function. This boundary condition is used to simulate the combustion chamber pressure.

2. Velocity Outlet: The velocity at the outlet is specified as a constant value or a time-dependent function. This boundary condition is used to simulate the exhaust velocity.

3. Temperature Outlet: The temperature at the outlet is specified as a constant value or a time-dependent function. This boundary condition is used to simulate the exhaust temperature.

Wall Boundary Conditions:

1. No-Slip Wall: The velocity at the wall is set to zero, simulating the no-slip condition.

2. Isothermal Wall: The temperature at the wall is specified as a constant value, simulating an isothermal boundary condition.

3. Adiabatic Wall: The heat flux at the wall is set to zero, simulating an adiabatic boundary condition.

4. Convective Wall: The heat flux at the wall is specified as a function of the temperature difference between the wall and the surrounding fluid.

Symmetry Boundary Conditions:

1. Symmetry Plane: The symmetry plane is used to simulate the axisymmetric or two-dimensional nature of the problem.

2. Periodic Boundary: The periodic boundary condition is used to simulate the periodic nature of the problem, such as in a rocket engine with multiple injectors.

Porous Wall Boundary Conditions:

1. Porous Wall Velocity: The velocity at the porous wall is specified as a function of the pressure drop across the wall.

2. Porous Wall Temperature: The temperature at the porous wall is specified as a function of the heat flux across the wall.

These boundary conditions are used to simulate the complex interactions between the porous injector, the combustion chamber, and the surrounding environment. By applying these boundary conditions, the numerical simulation can accurately capture the behavior of the porous injector and the combustion process.

1.2.2 Reynolds Number

The Reynolds number (Re) is a dimensionless quantity used in fluid mechanics to predict flow patterns in different fluid flow situations. It helps to determine whether a fluid flow is laminar (smooth and orderly) or turbulent (chaotic and irregular). A low Reynolds number indicates laminar flow, while a high value indicates turbulent flow. Reynolds number is crucial in engineering and fluid dynamics. The Reynolds number is given by the formula:

$$\mathbf{Re=\rho \cdot v \cdot L/ \mu} \quad \text{eq.1 1}$$

where:

- ρ is fluid density,
- v is the flow velocity,
- L is the characteristic length,
- μ is dynamic viscosity

1.2.3 Propellant Atomization:

The process of propellant atomization and mixing in the case of a porous injector is vastly different from coaxial injectors. The large fuel injection area results in exceedingly low injection velocities. Considering typical parameters of injection for coaxial injectors (e.g., Weber number velocity ,and momentum flux ratio),this should result in poor atomization and mixing quality leading to poor combustion performance. However, multiple hot-fire test campaigns showed a very good combustion performance of porous injectors, which is comparable to and exceeding that of coaxial injectors [8]. Obviously, an alternative process of atomization and mixing has to be the dominant mechanism for porous injectors.

In principle, the atomization mechanism of a porous injector is based on shear forces, just like the atomization process of a coaxial injector. In contrast to the later, the shear forces are not applied on the liquid jet at the injector tip but are a result of the processes taking place in the combustion chamber.

The proposed atomization mechanism is based on the experimental observation of a stable anchored flame at the LOX post tip. This flame produces the accelerating hot-gas environment responsible for a fast and efficient atomization process. It is not known if a lifted flame could support the observed efficient combustion in a similar way.

II. LITERATURE SURVEY

2.1 Motivation and Objective

Coaxial injectors represent the state of the art for injectors used ingas-/liquid-fueled rocket engines, especially for the high-performance cryogenic propellant combination LOX/H_2. Injector heads with shear-coaxial elements, when designed properly, offer good atomization performance, and therefore high combustion efficiency. Their behavior has been studied extensively over the last decades. The atomization process is based on the shear forces between a dense and slow liquid jet and a surrounding low-density fast gas stream.

These shear forces induce the breakup of the large diameter inner liquid jet into small-diameter droplets, which readily evaporate.

The atomization is usually characterized by dimensionless parameters such as 1) the Weber number, 2) the velocity ratio (VR), or 3) the momentum flux ratio between both propellants at the injector tip. Early investigations on shear-coaxial atomization behavior provided minimum values for these parameters to ensure stable and efficient combustion . Changes in the design of shear-coaxial injectors were introduced to increase both the combustion efficiency and the margin toward the onset of combustion instabilities. For example, recessing the central tube with respect to the faceplate to a certain degree results in increased combustion stability.

These favorable aspects of shear-coaxial injectors come with some disadvantages. Because the atomization process is controlled by the shear forces induced by the large velocity difference between both propellants, the injector geometry has to be tightly controlled, especially with respect to the concentricity of the coaxial propellant streams. Non concentricity of a coaxial injector element can result in severe losses in terms of atomization quality, and therefore combustion efficiency.

Another disadvantage is the off design behavior of the coaxial injector. This dependency stems from the fact that every coaxial injector is designed for a certain operating point. The geometrical layout of the injector elements ensures that important dimensionless parameters of injection do not fall below certain thresholds over the desired regime of operation. The dimensions of an individual injector element define the injection velocities for certain operating conditions, which in turn define the relevant dimensionless parameters.

This study focuses on the numerical simulation of a porous nozzle using computational fluid dynamics (CFD) techniques. Simulations are performed to analyze the flow characteristics. Residual pressure and fuel distribution within a porous structure. Multiple levels of porosity Nozzle shape and operating conditions are modeled to understand their influence on flow dynamics and mixing performance.

D. Suslov, J. Lux and O. J. Haidn findings of Altogether the new concept show several advantages over classic a lnjector design and is a very promising candidate for an application in an effusion cooled liquid rocket engine operated with LOX/H2 or LOX/CH4 at elevated pressures and realistic conditions.

Kim, Do-Hun & Shin, Jeung-Hwan & Lee, In-Chul & Koo, Jaye investigated the to improve the mixing and atomizing performance at the center region of the conventional coaxial shear injector spray, the concept of a coaxial porous injector was invented.

This novel injection concept for liquid rocket engines utilizes the Taylor-Culick flow in the cylindrical porous tube. The 2-dimensional injector, which can be converted in three injection configurations, was fabricated, and several cold flow tests using water-air simulant propellant was performed. The hydraulic characteristics and the effects of a gas flow condition on the spray pattern and the Sauter mean diameter (SMD) was analyzed for each configuration. The atomizing mechanism of coaxial porous injector was different with the coaxial shear injector, and it was explained by the momentum of the gas jet, which is injected normally against the center liquid column, and by the secondary disintegration at the wavy interface of liquid jet, which was generated at the recessed region. The SMD of 2D coaxial porous injector, which has higher gas momentum, was measured and it shows better atomizing performance at the center and outer side of spray than the 2D coaxial shear injector.

Zhukov, Victor & Suslov, Dmitry findings of Comparing the different numerical models the authors concluded that the wall heat flux predictions require “time accurate” simula-305tions (i.e. URANS or LES) due to the unsteadiness of the flow. In the present work the good agreement with the experimental results has been achieved us-ing RANS (Reynolds-averaged Navier–Stokes equations). In fact the chamber demonstrated a stable operation during the hot runs what means low fluctua-tions of flow parameters in the combustion chamber. In contrast to other com-310bustion chambers and the chamber modelled by Tucker et al. [20], the chamber with a porous injector head does not have significant stagnation regions which are potential sources of flow instabilities.

J. C. Deeken,∗ D. I. Suslov, M. Oschwald, and S. Schlechtriem investigated the The API concept offer several advantages compared to state of the art coaxial injectors, namely, a low sensitivity toward changes in injection conditions, a good off design performance ,as well as a more compact heat release zone in the combustor. The last point especially makes it a very attractive choice for an injector concept for expander-based engine cycles. Due to these favorable qualities, the Liquid Upper Stage Demonstrator Engine (LUMEN) currently under development will feature an API-type porous injector.

Lux, Johannes & Suslov, D. & Haidn, Oskar investigated the An alternative injection concept for the application in liquid propellant rocket engines has been successfully tested using two subscale rocket combustion chambers at pressures of up to 80 bar.

With the new concept liquid oxygen (LOX) enters the chamber through numerous small diameter tubes in a classical parallel showerhead configuration. The fuel is completely fed through a porous faceplate made from sinter bronze. Two propellant combinations have been used, LOX/H2 and LOX/CH4. The near injector zone has been investigated using an optically accessible combustion chamber.

III. METHODOLOGY

This project incorporates two different software tools and is structured into two main phases. The **design process** is carried out using **CATIA V5**, a 3D interactive modeling software. The second and more critical phase involves the aerodynamic of the designed model. This will be conducted in **ANSYS Fluent**, under well-defined boundary conditions to obtain accurate results.

3.1 Catia V5 Software:

Catia V5 is a widely used 3D CAD (computer-aided design) software developed by Dassault Systemes. It is a powerful tool for product design, engineering, and manufacturing, offering an intuitive interface and advanced modeling capabilities. Catia V5 is extensively utilized in industries such as aerospace, automotive, and industrial design due to its ability to handle complex geometries and multidisciplinary engineering tasks.

This software provides a parametric and feature-based approach to design, enabling users to create solid, surface, and sheet metal models with high precision. It supports assembly modeling, kinematic simulations, and structural analysis, making it a comprehensive solution for product development. Additionally, Catia V5 integrates seamlessly with other software tools, facilitating collaborative design and real-time modifications in large-scale projects.

With its user-friendly interface and extensive functionality, Catia V5 remains a preferred choice for engineers and designers looking to develop high-quality, innovative products efficiently.

3.1.1 Geometric Modelling

A 3D geometric model of the porous injector was created using CATIA. The model includes key structural features such as the injector's porous surface, inlet, and outlet regions. Special attention was given to defining the porous medium correctly to ensure accurate representation in simulations. The design was simplified by removing unnecessary small features that could increase computational cost while retaining essential characteristics for flow analysis. The final model was exported in a compatible format for meshing and numerical simulation.

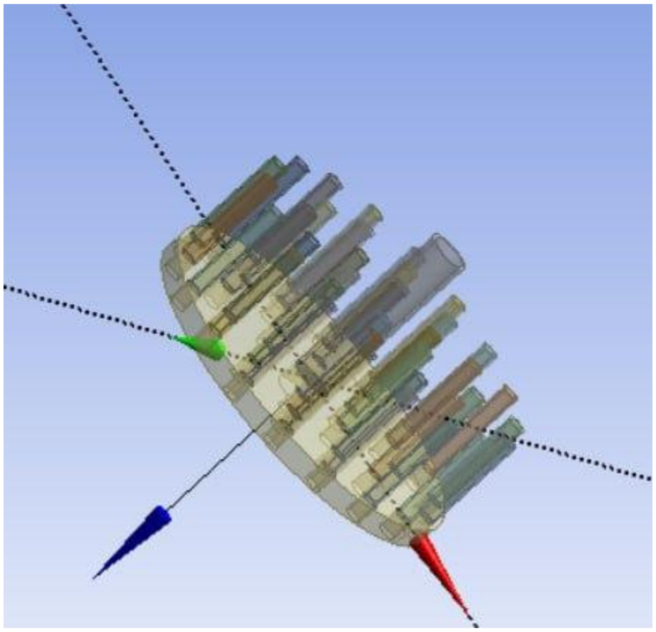


Figure 3.1.1 porous injector head design in CATIA

Design Dimensions and Process

Injector Outer Diameter: 2.6 mm

Injector Length: 15 mm

Porous Section Length: 3 mm

Porous Diameter: 50mm

Number of Pores: 36 (uniformly distributed)

Steps In Geometric Modeling:

Conceptual Design: The injector layout was sketched based on theoretical analysis and design requirements.

CAD Modeling: The design was created in CATIA, ensuring accurate dimensioning and feature placement.

Porous Structure Definition: The injector's porous section was modeled using a uniform distribution of pores, with dimensions derived from permeability studies.

Optimization: Small features that could increase computational cost were removed while maintaining critical geometric aspects.

Exporting the Model: The final model was saved in STEP/IGES format for meshing and CFD simulation.

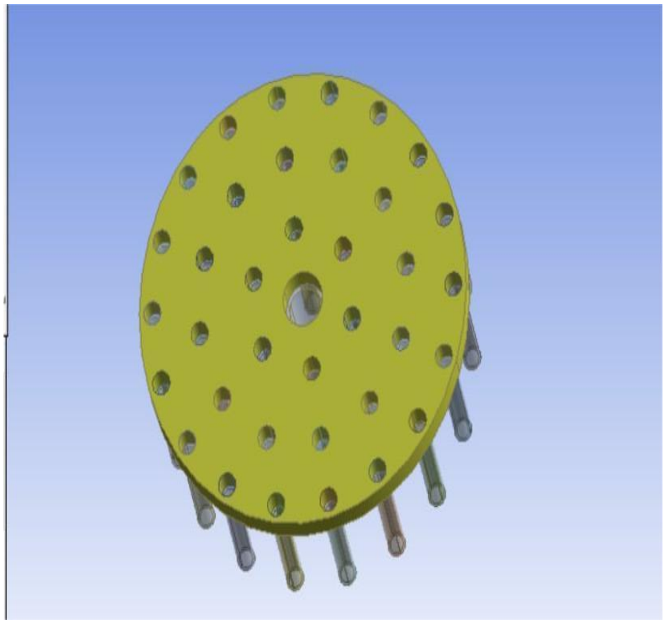
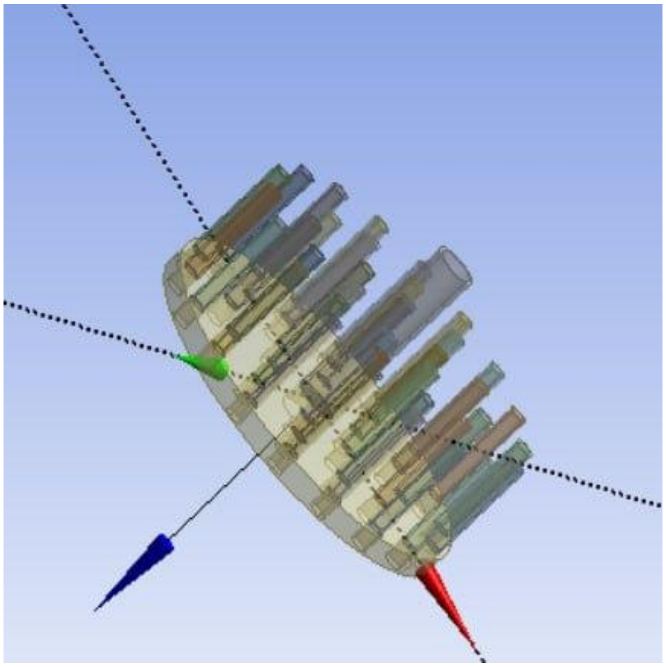


Figure 3.1.3: porous injector head face plate with hole being in circular pattern



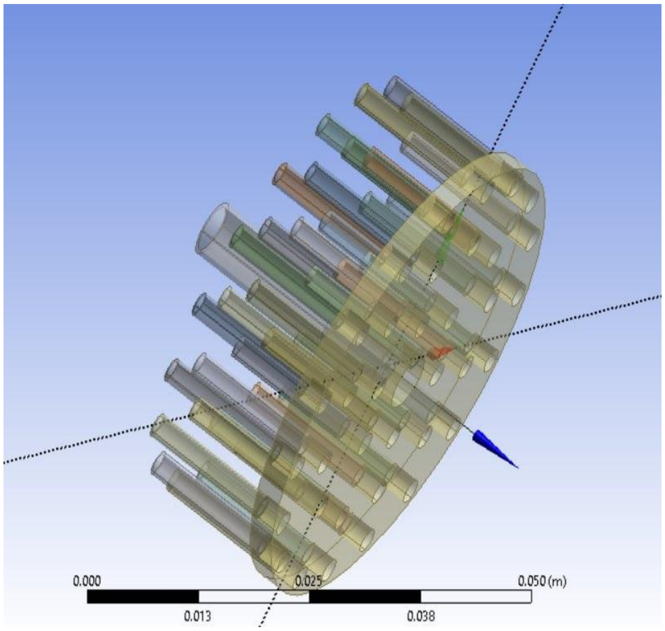
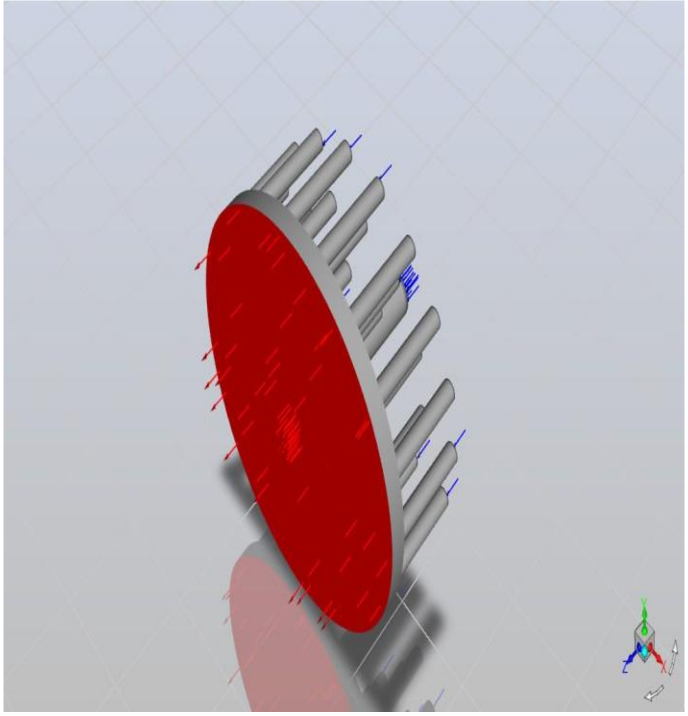


Figure 3.1.4 multi view of porous injector head

3.2 ANSYS Software:



ANSYS is a widely used engineering simulation software developed by ANSYS Inc., designed to analyze and solve complex structural, fluid, thermal, and electromagnetic problems. It provides advanced computational tools that enable engineers and researchers to simulate real-world physical conditions, helping to optimize designs before manufacturing.

One of its key features is the use of the finite element method (FEM) and computational fluid dynamics (CFD) to model and analyze various engineering scenarios. ANSYS Fluent, a specialized module within the software, is commonly used for fluid flow and heat transfer simulations, making it ideal for aerodynamic studies.

The software supports a range of industries, including aerospace, automotive, electronics, and energy, allowing users to conduct structural integrity checks, thermal performance analysis, and flow simulations. With its user-friendly interface and robust solver capabilities, ANSYS plays a crucial role in modern engineering by reducing development time and improving the accuracy of designs.

3.2.1 Modelling

After the geometric model was created in CATIA, it was imported into ANSYS for computational fluid dynamics (CFD) analysis. The model was checked for errors, such as gaps, overlapping surfaces, or non-manifold geometries, which could affect simulation accuracy. A fluid domain was created around the injector to represent the flow environment accurately. The porous section was defined with appropriate material properties to simulate realistic flow resistance.

The ANSYS DesignModeler tool was used to prepare the geometry for analysis, including defining boundary conditions, fluid regions, and flow paths. Boolean operations were performed to separate the solid and fluid domains where necessary. Named selections were assigned to the inlet, outlet, walls, and porous section for easier meshing and boundary condition application in the solver.

3.2.2 Meshing:

The geometric model was imported into a computational fluid dynamics (CFD) software, where a high-quality mesh was generated. A structured or hybrid meshing approach was employed to capture flow characteristics accurately while balancing computational cost. To ensure solution accuracy, a mesh independence study was conducted. Boundary layer refinement was applied to capture near-wall effects critical to turbulent flow simulation. The mesh quality was assessed based on metrics such as skewness, aspect ratio, and orthogonality to ensure numerical stability and reliable results.

The meshing process involved multiple steps, beginning with the selection of an appropriate mesh type. A hybrid mesh was preferred, combining structured elements in the core flow regions and unstructured elements in complex porous sections. The injector walls and porous section were refined with a high-density mesh to capture intricate flow features and pressure gradients accurately. A boundary layer mesh was incorporated to resolve viscous effects near walls, ensuring proper turbulence modeling. Adaptive mesh refinement (AMR) was utilized to enhance resolution in regions with high velocity or pressure gradients. A mesh independence study was performed by generating multiple mesh resolutions and comparing results to ensure that further refinement did not significantly affect the accuracy of the solution. The final mesh quality was evaluated based on skewness, aspect ratio, and orthogonality, with all parameters maintained within recommended limits to ensure numerical stability and convergence.

Additionally, specialized meshing techniques were employed to ensure accurate simulation of flow through the porous structure. A conformal mesh approach was used to align elements precisely at the interface between solid and porous regions, minimizing numerical diffusion. A finer mesh was applied in the porous section to resolve microscopic flow behavior while coarser elements were used in the bulk flow regions to optimize computational efficiency. Transitioning between mesh densities was carefully handled to prevent abrupt changes that could impact solution accuracy. Mesh refinement techniques such as inflation layers were applied near walls to improve resolution of boundary layer effects. The final mesh consisted of approximately 2-5 million elements, ensuring a good balance between accuracy and computational feasibility.

3.3 Boundary Conditions

Boundary conditions were defined based on theoretical assumptions and experimental data. At the inlet, a specified velocity or mass flow rate was applied, while the outlet was defined with either pressure or ambient conditions. No-slip conditions were enforced on the walls, and the porous zone was characterized using Darcy’s law or empirical permeability data. A steady or transient solver was chosen depending on the flow characteristics, and convergence criteria were set based on residuals and key flow parameters.

The numerical model was executed under different operating conditions to evaluate its performance. The simulation results were validated by comparing them with available experimental data or literature sources.

A sensitivity analysis was performed to understand the effects of mesh refinement, solver settings, and boundary conditions on the results. Convergence was ensured by monitoring the stability of key parameters over successive iterations.

The simulation is based on the Navier-Stokes equations, incorporating the continuity equation for mass conservation, momentum equations for fluid flow dynamics, and the energy equation when thermal effects are considered. A porous media model was implemented to simulate fluid interactions within the injector structure accurately. To capture flow variations and turbulence, an appropriate turbulence model such as k-ε or k-ω SST was applied.

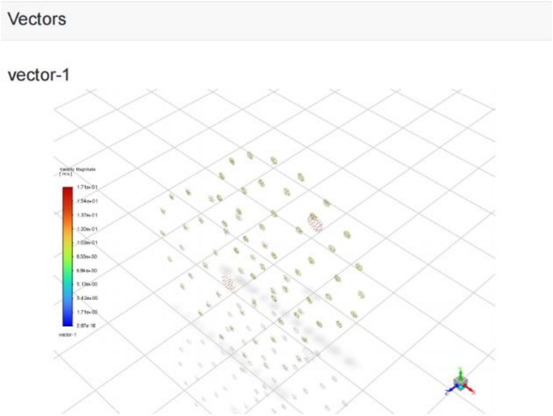
Post-processing involved analyzing the simulation results using contour plots, velocity streamlines, pressure distributions, and injector performance metrics. The effect of porosity on flow distribution, pressure drop, and injector efficiency was evaluated. Insights gained from the analysis were used to identify potential improvements in the injector design.

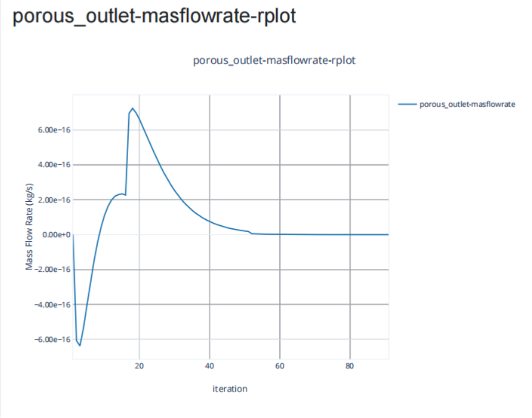
IV. RESULTS

4.1 Velocity Distribution

The velocity contours obtained from the ANSYS Fluent simulation show the variation of fluid velocity within the porous injector. It is observed that the velocity inside the porous region is relatively low due to the resistance offered by the porous medium.

As the fluid exits through the pores, multiple localized high-velocity jets are formed. These jets enhance the mixing process by increasing the interaction between the injected fluid and the surrounding medium. Compared to conventional injectors, the flow is more uniformly distributed, reducing flow irregularities and improving performance.

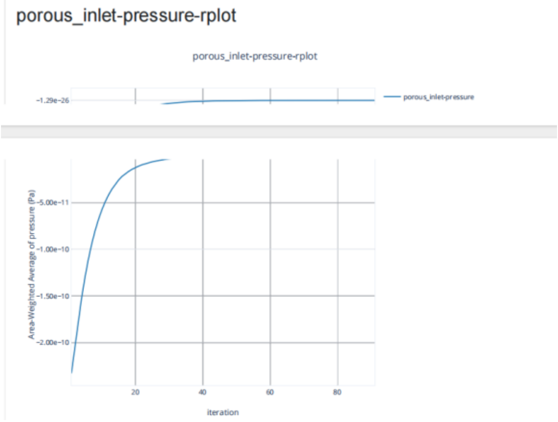


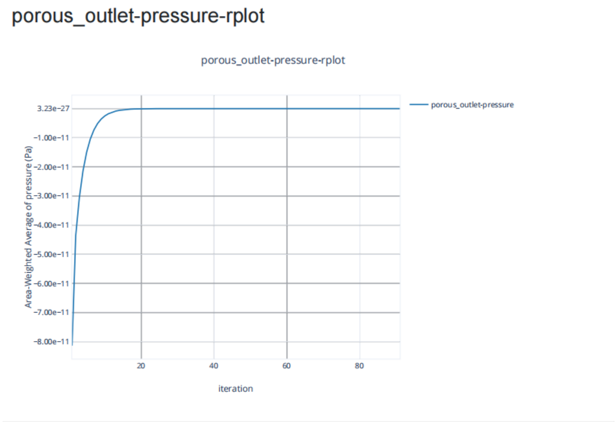


4.2 Pressure Distribution

The pressure contours indicate a gradual decrease in pressure from the inlet to the outlet of the injector. A significant pressure drop is observed across the porous region due to the permeability resistance of the porous material.

This pressure drop plays a crucial role in controlling the flow rate and ensuring uniform distribution of the fluid across all pores. The results confirm that the porous injector acts as an effective flow regulator, preventing sudden pressure fluctuations.



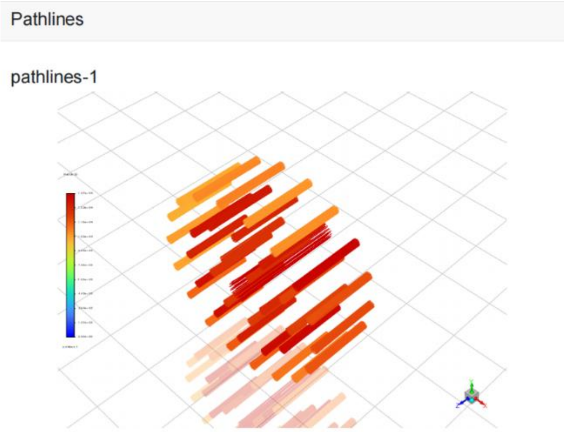


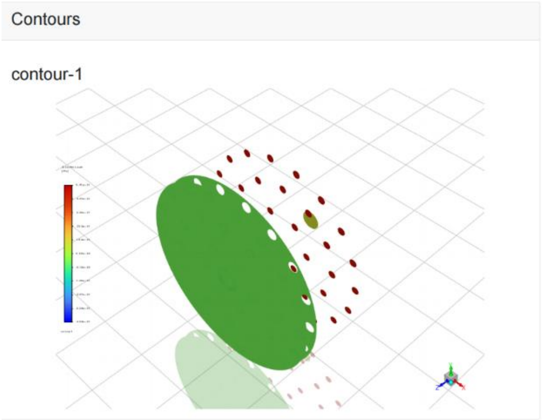
4.3 Flow Behavior in Porous Region

The flow through the porous region exhibits complex interaction between the fluid and the solid matrix. The fluid is distributed through multiple small pores, which reduces the formation of large concentrated jets.

This distributed flow pattern leads to:

- Improved mixing efficiency
- Reduced turbulence intensity at the inlet
- Better stability in flow behavior
- The results clearly indicate that the porous structure helps in achieving uniform flow distribution.





4.4 Effect of Porosity

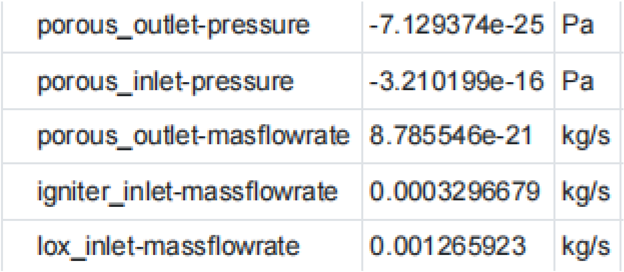
The simulation results show that porosity significantly affects the flow characteristics. Higher porosity allows easier flow of fluid, resulting in higher velocities and lower pressure drop.

On the other hand, lower porosity increases resistance, leading to higher pressure drop and reduced velocity. Therefore, selecting an optimal porosity is essential to balance pressure loss and mixing efficiency.

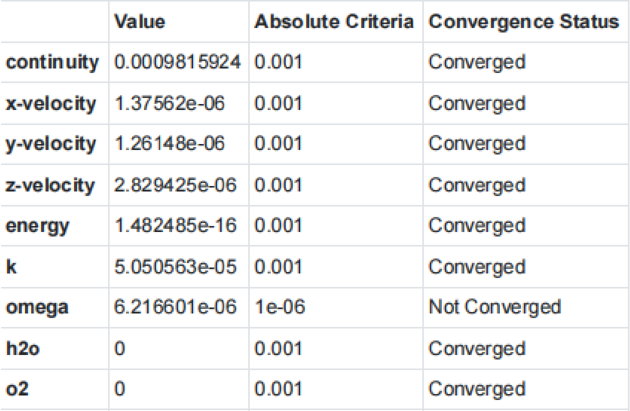
4.5 Discussion of Results

From the overall analysis, it is observed that the porous injector provides better flow uniformity compared to traditional injectors. The multiple pore structure distributes the flow evenly and reduces localized high-velocity regions.

The pressure drop across the porous medium ensures controlled flow, while the velocity distribution enhances atomization and mixing. These characteristics contribute to improved combustion efficiency and stability.



porous_outlet-pressure	-7.129374e-25	Pa
porous_inlet-pressure	-3.210199e-16	Pa
porous_outlet-masflowrate	8.785546e-21	kg/s
igniter_inlet-massflowrate	0.0003296679	kg/s
lox_inlet-massflowrate	0.001265923	kg/s



	Value	Absolute Criteria	Convergence Status
continuity	0.0009815924	0.001	Converged
x-velocity	1.37562e-06	0.001	Converged
y-velocity	1.26148e-06	0.001	Converged
z-velocity	2.829425e-06	0.001	Converged
energy	1.482485e-16	0.001	Converged
k	5.050563e-05	0.001	Converged
omega	6.216601e-06	1e-06	Not Converged
h2o	0	0.001	Converged
o2	0	0.001	Converged

V. CONCLUSION

The present study focused on the numerical simulation of a porous injector in a liquid rocket engine using ANSYS Fluent. A three-dimensional model was developed and analyzed to understand the flow behavior, pressure distribution, and mixing characteristics.

The results show that the porous injector provides uniform flow distribution due to the presence of multiple pores. The velocity distribution indicates the formation of small jets at the outlet, which enhances mixing and atomization. The pressure analysis reveals a controlled pressure drop across the porous region, which plays a key role in regulating the flow.

Compared to conventional injectors, porous injectors demonstrate improved performance in terms of flow stability, mixing efficiency, and adaptability to varying operating conditions. The distributed injection mechanism reduces flow irregularities and enhances combustion characteristics.

Overall, the study confirms that porous injectors are a promising alternative for modern liquid rocket engines.

A. Future Scope

- Experimental validation of simulation results
- Optimization of pore size and distribution
- Analysis with different propellants
- Transient and combustion simulations

REFERENCES

[1] Kumakawa,A.,Sato,K.,Sasaki,M.,Tamura,H.,andHashimoto,T.,"Hotgassideheat transfer characteristics of LOX/H2andLOX/HC type propellants, "Proceedings of the Sixteenth International Symposiumon Space TechnologyandScience,Sapporo,Japan,1988

[2] Tamura,H.,Ono,F.,Kumakawa,A.,andYatsuyanagi,N.,"LOX/Methane Staged Combustion Rocket Combustor Investigation ,"23rdAIAA/ASME/SAE/ASEE Joint Propulsion Conference ,American Institute of Aeronautics andAstronautics,SanDiego,CA,June-July1987,AIAA1987-1856

[3] Tamura, H., Sakamoto, H., Sasaki, M., Takahashi, M., Tomita, T., and Mayer, W., "An experimental study on the stability characteristics of the LOX/methane rocket combustor," 31th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, American Institute of Aeronautics and Astronautics, San Diego, CA, July 1995, AIAA 95-2359.

[4] Wheeler, D.B.andKirby, F. M., "High-Pressure LOX/CH4Injector Program," Tech. rep., Rockwell International, Rocketdyne Division, 1979, NASA-CR-161342.

[5] Carlile, J. A., and Quentmeyer, R. J., "An Experimental Investigation of High-Aspect-Ratio Cooling Passages," NASA TM-105679, 1992. doi:10.2514/6.1992-3154

[6] Pavli, A. J., Kacynski, K. J., and Smith, T. A., "Experimental Thrust Performance of a High-Area-Ratio Rocket Nozzle," NASA TP-2720, 1987.

[7] Kahrs, J., and Corbett, A., "Propellant Injection Through Porous Media," AIAA 3rd Propulsion Joint Specialist Conference, AIAA Paper 1967-0463, 1967. doi:10.2514/6.1967-463