

FOSSEE Summer Fellowship

Development of an Overset Mesh Framework for a Single-Orifice Rotary Slinger Atomizer Using OpenFOAM

Manikandan Padmanaban¹, Evan Fernandes², Arnab Chakrabarty³, Dhiraj Garg⁴

¹Department of Applied Mechanics and Biomedical Engineering, Indian Institute of Technology Madras

²FOSSEE, Indian Institute of Technology Bombay

³TCS Research and Innovation

⁴Department of Chemical Engineering, Shiv Nadar University

Abstract

Rotary slinger atomizers are widely used in aerospace propulsion and gas turbine combustors due to their ability to generate fine fuel droplets through centrifugal atomization. Numerical simulation of these systems is challenging because the rotating geometry requires an efficient mesh handling strategy while maintaining computational accuracy. The overset mesh technique offers an effective solution by allowing independent motion of rotating and stationary mesh regions without mesh deformation. This work presents the development of an overset mesh framework for a single-orifice rotary slinger atomizer using OpenFOAM. A two-dimensional computational model consisting of a rotating slinger disc with a single discharge orifice and a stationary background domain was developed. The rotor geometry was created using Autodesk Fusion 360, and a body-fitted computational mesh was generated and verified to satisfy the required mesh quality criteria. The final computational domain consists of a stationary background mesh combined with a rotating overset mesh, providing a robust framework for rotating flow simulations. The developed methodology establishes a reliable preprocessing workflow for rotary slinger atomizers and can be readily extended to more complex geometries. Future work will utilize the developed framework with the `interDyMFoam` solver and the Volume of Fluid (VOF) method to investigate liquid discharge, primary droplet breakup, and spray evolution under different operating conditions.

1 Introduction

Atomization is the process of disintegrating a bulk liquid into fine droplets and is widely employed in combustion systems, spray drying, painting, agricultural spraying, and pharmaceutical applications. In gas turbine engines, efficient atomization is essential for rapid fuel evaporation, improved

fuel-air mixing, stable combustion, and reduced pollutant emissions. The droplet size distribution generated by an atomizer strongly influences combustion efficiency and overall engine performance. Among various atomization technologies, rotary atomizers utilize the rotational kinetic energy of the liquid to produce atomization. In these systems, liquid is supplied to a rotating component such as a disc or cup, where centrifugal forces drive the liquid radially outward. As the liquid exits the rotating surface, aerodynamic interaction with the surrounding air causes the liquid structures to break into droplets. Rotary atomizers are attractive because they can atomize a wide range of liquids and are extensively used in industrial spray processes.

A special class of rotary atomizers known as rotary slinger atomizers is widely employed in small gas turbine engines. A slinger atomizer consists of a rotating disc containing multiple orifices located on its peripheral surface. Fuel is supplied to the rotating disc and is ejected through these orifices under the action of centrifugal forces. Compared with conventional pressure-swirl atomizers, slinger atomizers offer several advantages, including simpler construction, reduced dependence on high-pressure fuel pumps, improved fuel flexibility, and suitability for compact engine configurations. Furthermore, the rotational speed and liquid flow rate can be controlled independently, providing additional flexibility in atomization performance. Several experimental investigations have been conducted to understand the breakup and spray characteristics of slinger atomizers. Early studies focused on the influence of rotational speed, liquid flow rate, orifice size, and disc geometry on droplet size and breakup behavior. Dahm et al. identified subcritical and supercritical breakup modes and developed a theoretical model for predicting liquid film thickness inside the slinger orifice. Later investigations by Choi and co-workers explored the effects of hole size, hole arrangement, and rotational speed on spray characteristics. More recent studies by Rezayat et al. reported the existence of Coriolis-induced liquid accumulation inside the discharge channels and demonstrated the influence of liquid Weber number and aerodynamic Weber number on breakup regimes and droplet size distributions.

Despite these advances, the fundamental understanding of liquid flow inside the rotating orifice remains limited. Experimental measurements inside the narrow rotating channels are challenging because parameters such as liquid film thickness, liquid span, velocity distribution, and local pressure fields cannot be directly measured. Existing analytical models often neglect either the Coriolis force or surface tension effects, although experimental observations clearly indicate that both mechanisms strongly influence liquid distribution and breakup behavior. Consequently, a detailed numerical investigation is required to complement experimental observations and provide deeper insight into the internal flow physics. The experimental study by Sahu et al. demonstrated that liquid accumulates preferentially on the side opposite to the direction of rotation due to the Coriolis force. Their visualization results revealed that increasing rotational speed reduces liquid film thickness and enhances atomization, whereas increasing liquid flow rate primarily increases the liquid span inside the orifice. The study further established relationships between breakup modes, Coriolis Bond number, and droplet size evolution.

Therefore, the present internship work aims to develop a Computational Fluid Dynamics (CFD) framework for simulating the liquid flow and breakup behavior inside a rotary slinger atomizer. The numerical model is based on OpenFOAM and incorporates a rotating injector geometry with a single discharge orifice. An overset mesh strategy is employed to accurately represent the rotating domain while maintaining computational efficiency. The objective is to predict the liquid distribution inside the rotating channel, investigate the influence of centrifugal and Coriolis forces, and establish a numerical methodology capable of supporting future studies on fuel atomization and

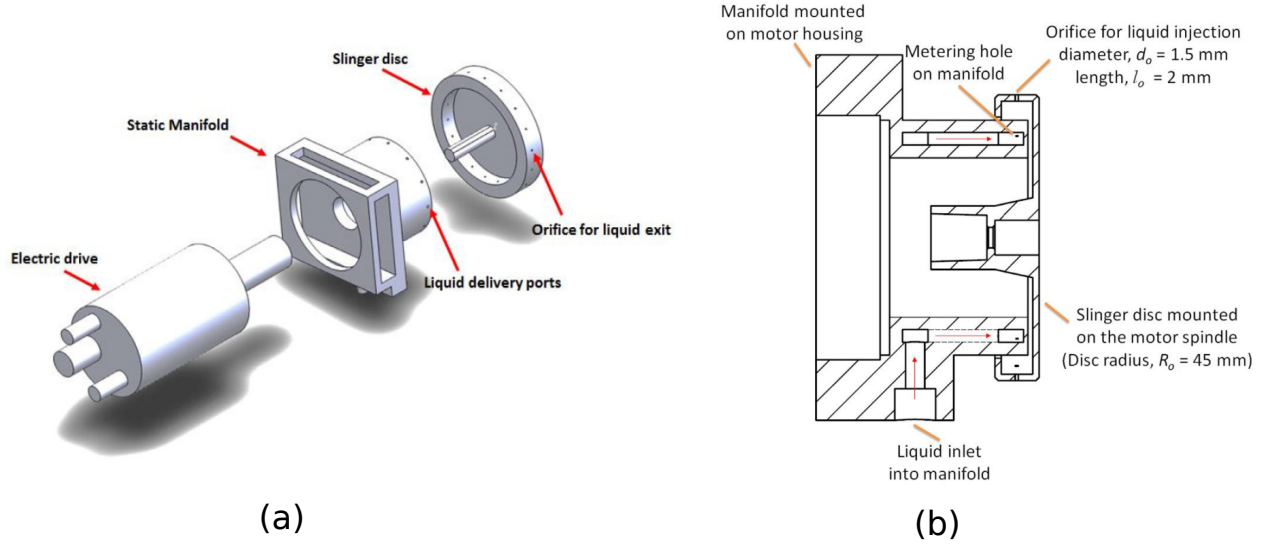


Figure 1: Slinger atomizer assembly (a) Different components including the electric drive, liquid delivery manifold (static) and the rotating slinger disc. (b) Cross-sectional view [1].

spray formation in high-speed rotary atomizers.

2 Problem Statement

Rotary slinger atomizers are widely used in small gas turbine engines due to their simple design, fuel flexibility, and ability to generate fine sprays without requiring high fuel injection pressures. Experimental investigations have shown that the liquid flow within the rotating orifice is strongly influenced by centrifugal and Coriolis forces, resulting in asymmetric liquid distribution, complex breakup behavior, and variations in droplet size. However, direct measurement of important flow parameters such as liquid film thickness, liquid span, velocity distribution, pressure field, and interfacial dynamics inside the rotating orifice is extremely difficult because of the small geometric dimensions and high rotational speeds involved. Previous experimental studies have primarily focused on visualizing the liquid breakup process and measuring the resulting droplet characteristics. Although these investigations provide valuable information regarding atomization performance, they do not offer detailed insight into the internal flow physics governing the liquid transport inside the rotating slinger channel. Consequently, a numerical framework capable of resolving the gas-liquid interface and capturing the effects of centrifugal and Coriolis forces is required to improve the understanding of the atomization process.

The objective of the present internship work is to develop a computational fluid dynamics (CFD) model of a rotary slinger atomizer using OpenFOAM in Fig 1. To reduce computational cost while retaining the essential flow physics, a two-dimensional slice of the experimental geometry is considered. The numerical domain consists of a stationary liquid delivery manifold (stator) and a rotating slinger disc containing a single discharge orifice (rotor). An overset mesh methodology is employed to enable relative motion between the stationary and rotating domains while maintaining mesh quality during simulation. The developed numerical model aims to investigate

the liquid distribution inside the rotating orifice, analyze the influence of rotational speed on liquid film formation, evaluate the role of Coriolis-induced liquid accumulation, and establish a computational framework that can be extended to predict primary breakup and spray formation in rotary slinger atomizers.

3 Governing Equations and Models

The incompressible two-phase flow is solved using the Volume of Fluid (VOF) method, where a single set of governing equations is solved throughout the computational domain. The interface between the liquid and gas phases is captured using the volume fraction field, α .

The continuity equation is given by

$$\nabla \cdot \mathbf{U} = 0, \quad (1)$$

where $\mathbf{U}$ is the velocity vector.

The momentum equation is

$$\frac{\partial(\rho\mathbf{U})}{\partial t} + \nabla \cdot (\rho\mathbf{U}\mathbf{U}) = -\nabla p + \nabla \cdot [\mu (\nabla\mathbf{U} + (\nabla\mathbf{U})^T)] + \rho\mathbf{g} + \sigma\kappa\nabla\alpha, \quad (2)$$

where ρ is the mixture density, μ is the dynamic viscosity, p is the pressure, $\mathbf{g}$ is the gravitational acceleration, σ is the surface tension coefficient, and κ is the interface curvature,

$$\kappa = -\nabla \cdot \left(\frac{\nabla\alpha}{|\nabla\alpha|} \right). \quad (3)$$

The transport equation for the liquid volume fraction is

$$\frac{\partial\alpha}{\partial t} + \nabla \cdot (\alpha\mathbf{U}) + \nabla \cdot [\alpha(1-\alpha)\mathbf{U}_r] = 0, \quad (4)$$

where $\mathbf{U}_r$ is the interface compression velocity used to maintain a sharp interface.

The mixture density and viscosity are evaluated as

$$\rho = \alpha\rho_l + (1-\alpha)\rho_g, \quad (5)$$

$$\mu = \alpha\mu_l + (1-\alpha)\mu_g, \quad (6)$$

where ρ_l and ρ_g are the liquid and gas densities, while μ_l and μ_g are the corresponding dynamic viscosities. The volume fraction satisfies $0 \leq \alpha \leq 1$, where $\alpha = 1$ denotes the liquid phase, $\alpha = 0$ denotes the gas phase, and $0 < \alpha < 1$ represents the interface between the two phases.

4 Simulation Methodology

4.1 Geometry Development

A two-dimensional slice of the rotary slinger atomizer was considered in the present study to reduce computational cost while retaining the essential flow physics. The computational domain consists

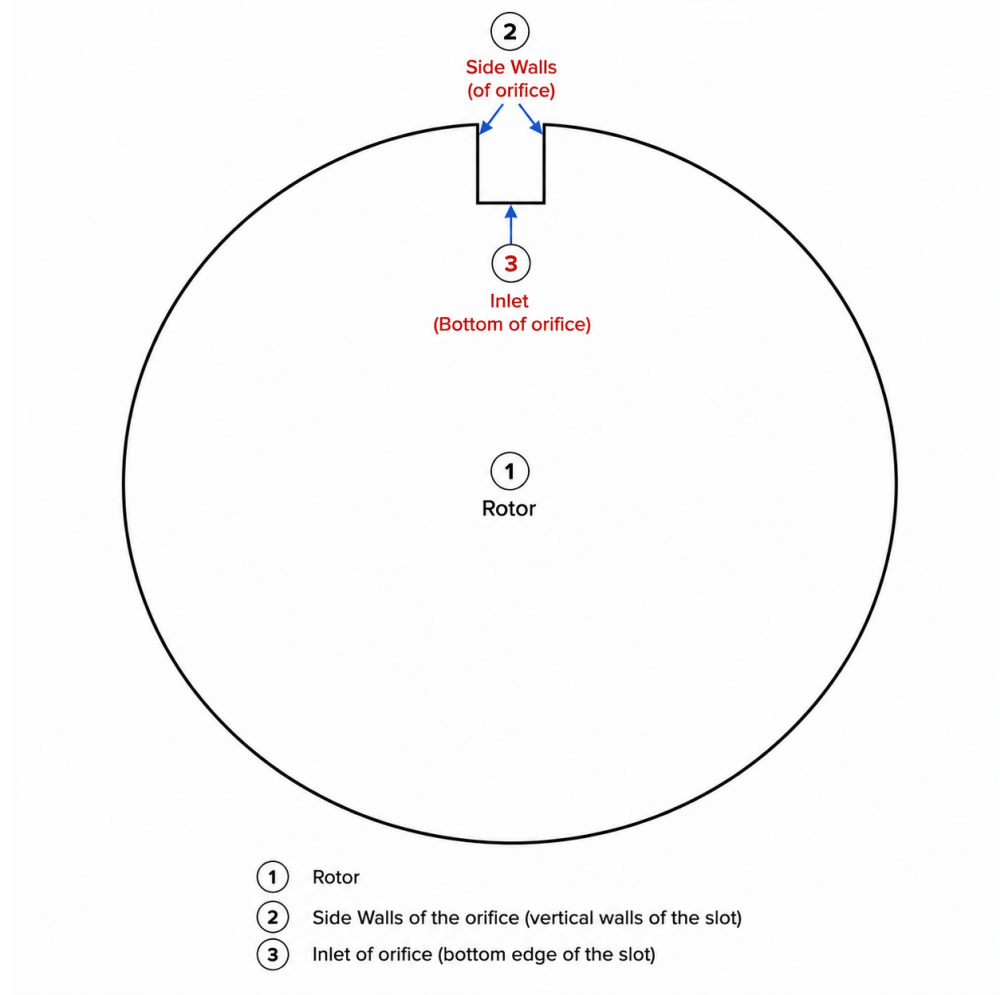


Figure 2: Geometry of the single-orifice rotary slinger atomizer showing the rotor, liquid inlet, single discharge orifice, and side-wall boundaries used for mesh generation.

of a stationary background region and a rotating slinger disc containing a single discharge orifice. The rotor geometry was developed in Autodesk Fusion 360 based on the dimensions of the experimental configuration and exported as STL surface files representing the rotor wall, inlet, side walls, and front and back surfaces in Figure 2.

4.2 Mesh Generation Framework

The computational mesh was generated entirely using OpenFOAM utilities following the workflow illustrated in Figures 3 and 4. A structured background mesh was first generated using the `blockMesh` utility to define the computational domain. The geometric feature edges of the STL model were extracted using `surfaceFeatureExtract`, after which `snappyHexMesh` was employed to generate a body-fitted mesh with local refinement around the rotor and orifice.

To obtain a two-dimensional computational domain, the front and back boundary patches were modified to empty boundary conditions. The mesh was subsequently processed using `flattenMesh`,

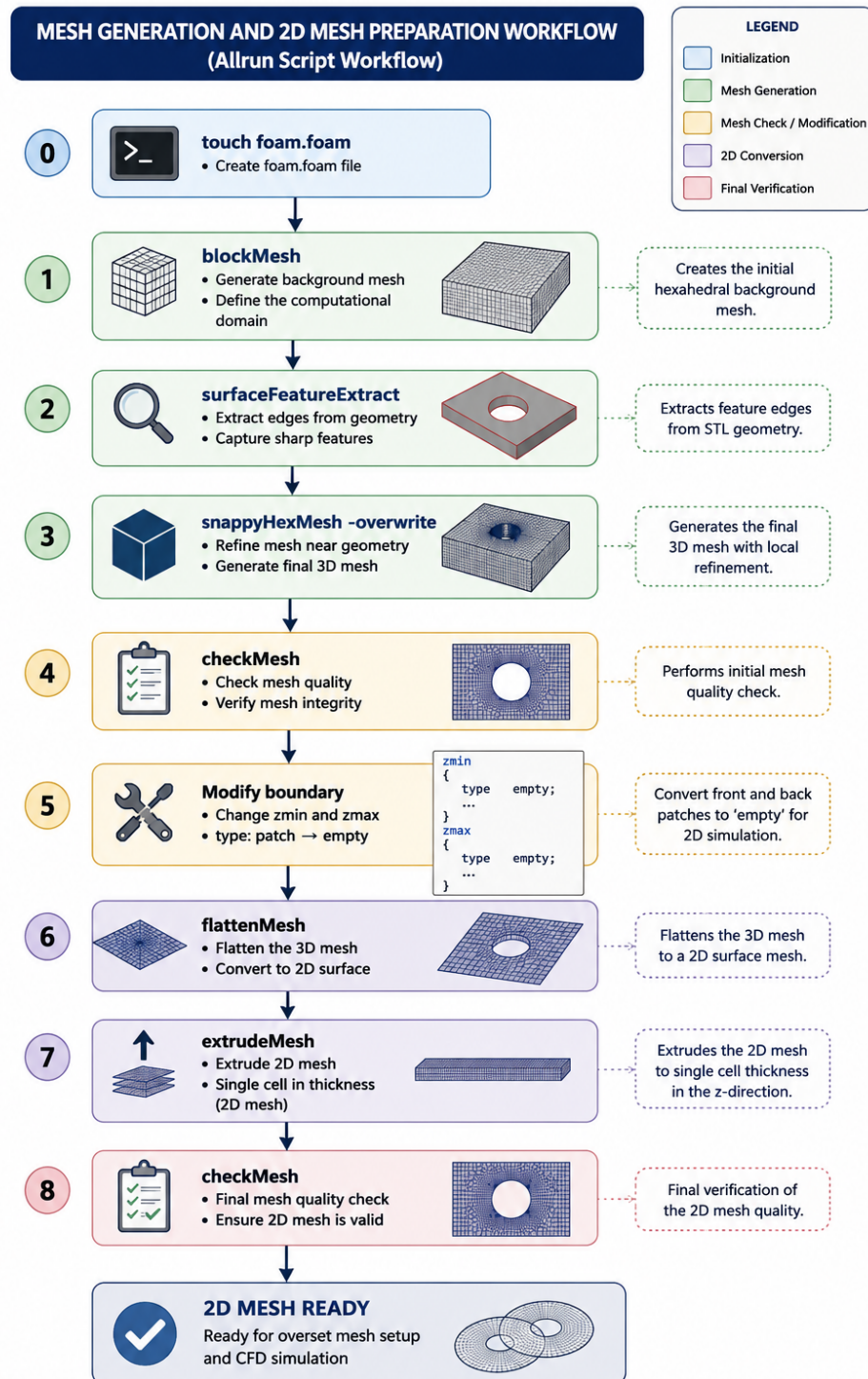


Figure 3: Background mesh preparation workflow for overset mesh development.

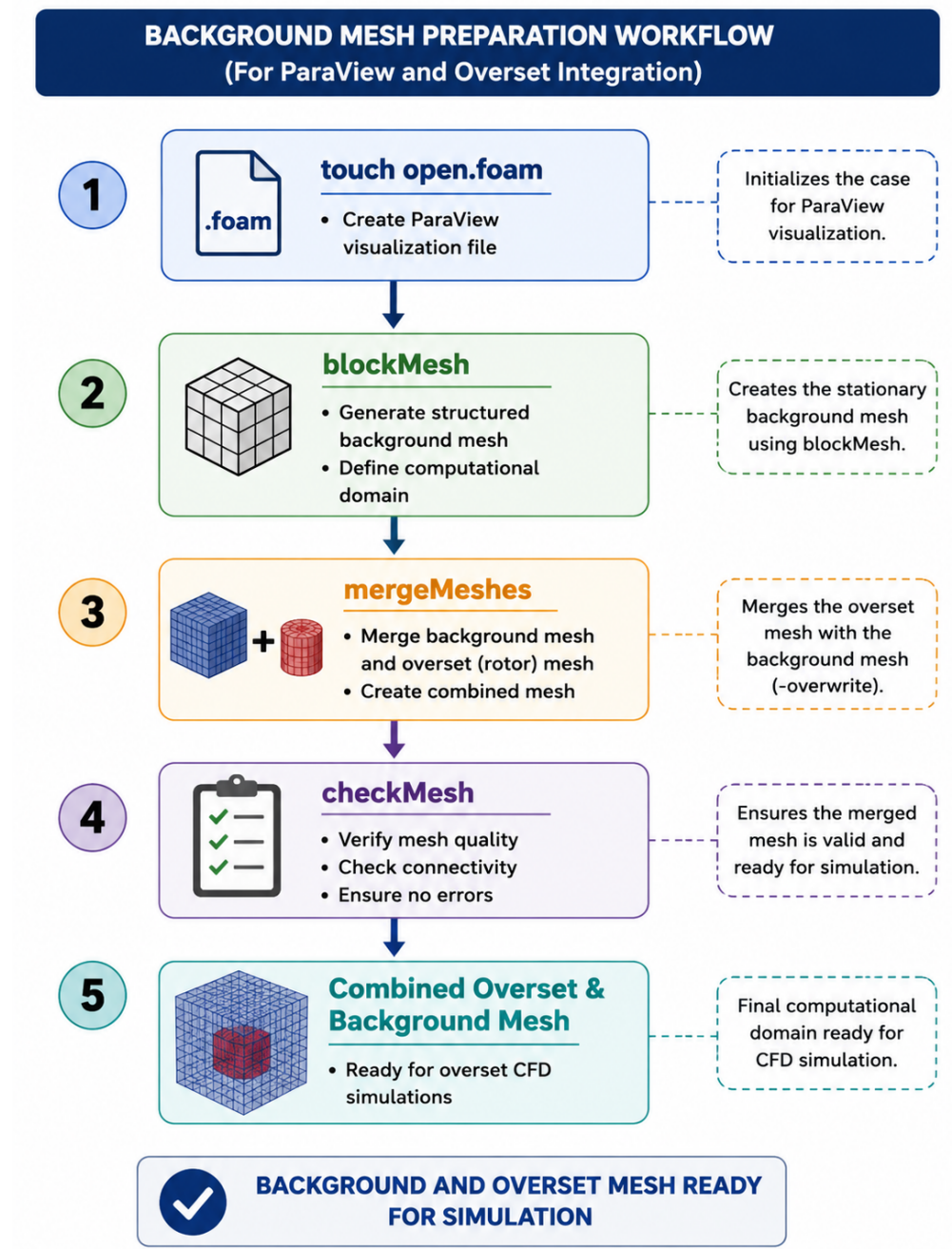


Figure 4: Complete mesh generation workflow adopted in the present study.

followed by `extrudeMesh` to generate a single-cell-thick mesh in the spanwise direction. Finally, the rotating overset mesh was merged with the stationary background mesh using `mergeMeshes`. The generated computational mesh was verified using the `checkMesh` utility before being employed for numerical simulations.

4.3 Initial and Boundary Conditions

The numerical simulations were performed using the incompressible two-phase flow solver `overInterDyMFoam`. Appropriate velocity, pressure, and phase fraction boundary conditions were prescribed at the inlet, outlet, rotor wall, stationary wall, and overset interfaces. The rotating region was defined using the overset mesh approach, while the outer computational domain remained stationary throughout the simulation.

5 Results and Discussion

5.1 Generated Computational Mesh

Figure 5 presents the final computational mesh generated for the single-orifice rotary slinger atomizer. Local mesh refinement was achieved near the rotor wall and discharge orifice using `snappyHexMesh`, while comparatively coarser cells were retained in the far-field region to reduce the computational cost. The adopted meshing strategy provides adequate geometric resolution near the rotating components while maintaining an efficient overall cell count.

Figure 6 presents the final computational mesh obtained after merging the stationary background mesh with the rotating overset mesh. The background mesh represents the stationary computational domain, while the overset mesh encloses the rotating slinger rotor containing the single discharge orifice. The two mesh regions overlap to facilitate data interpolation across the overset interface, enabling independent rotor motion without mesh deformation. The merged mesh preserves the geometric fidelity of the rotor while maintaining a structured background mesh, providing a robust computational framework for transient simulations. The successful integration of the background and overset meshes demonstrates the capability of the developed methodology to model rotating components efficiently and serves as the foundation for future VOF simulations of liquid breakup and spray evolution in the rotary slinger atomizer.

5.2 Overset Mesh Configuration

Figure 7 presents the computational mesh after applying the `flattenMesh` utility. This operation collapses the three-dimensional mesh along the spanwise direction, producing a planar two-dimensional mesh while preserving the geometric features of the rotor and the single discharge orifice. The flattened mesh significantly reduces the computational complexity by eliminating unnecessary thickness in the out-of-plane direction. The resulting mesh serves as an intermediate stage in the mesh preparation process prior to extrusion and subsequent overset mesh development. Figure 8 shows the final two-dimensional mesh obtained after applying the `extrudeMesh` utility. A single layer of cells was extruded in the spanwise direction to generate a computational mesh with one-cell thickness, satisfying the requirements for two-dimensional finite volume simulations in

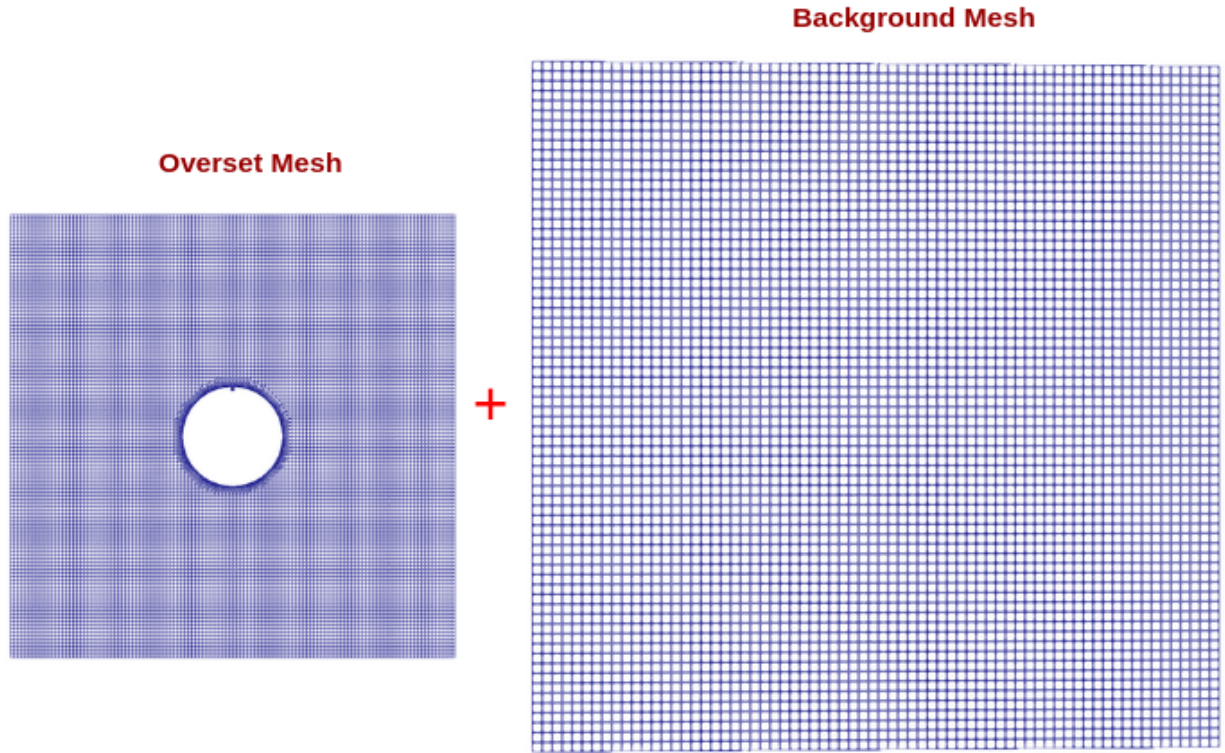


Figure 5: Computational mesh with Overset Mesh and Background mesh.

OpenFOAM. The extrusion process preserves the mesh topology and improves mesh consistency compared with the flattened mesh. The generated mesh successfully passed all `checkMesh` quality criteria and provides the computational domain used for the subsequent overset mesh generation and transient two-phase flow simulations.

5.3 Mesh Quality Assessment

The quality of the generated mesh was evaluated using the OpenFOAM utility `checkMesh`. Table 1 summarizes the principal mesh quality parameters. The generated mesh satisfied all OpenFOAM quality criteria. The maximum aspect ratio and non-orthogonality were well below the recommended limits for stable finite-volume simulations. No negative cell volumes, illegal faces, or connectivity errors were detected. The negligible cell openness and zero coupled point matching error further confirm the geometric consistency of the computational mesh. These results demonstrate that the generated mesh is suitable for overset mesh simulations of the rotary slinger atomizer.

Figure 9 presents the directory structure adopted for the OpenFOAM case used in the present study. The computational framework is organized into separate directories for the stationary background mesh and the rotating overset mesh, allowing independent mesh generation and pre-processing. Each mesh directory contains the `0.orig`, `constant`, and `system` folders together with

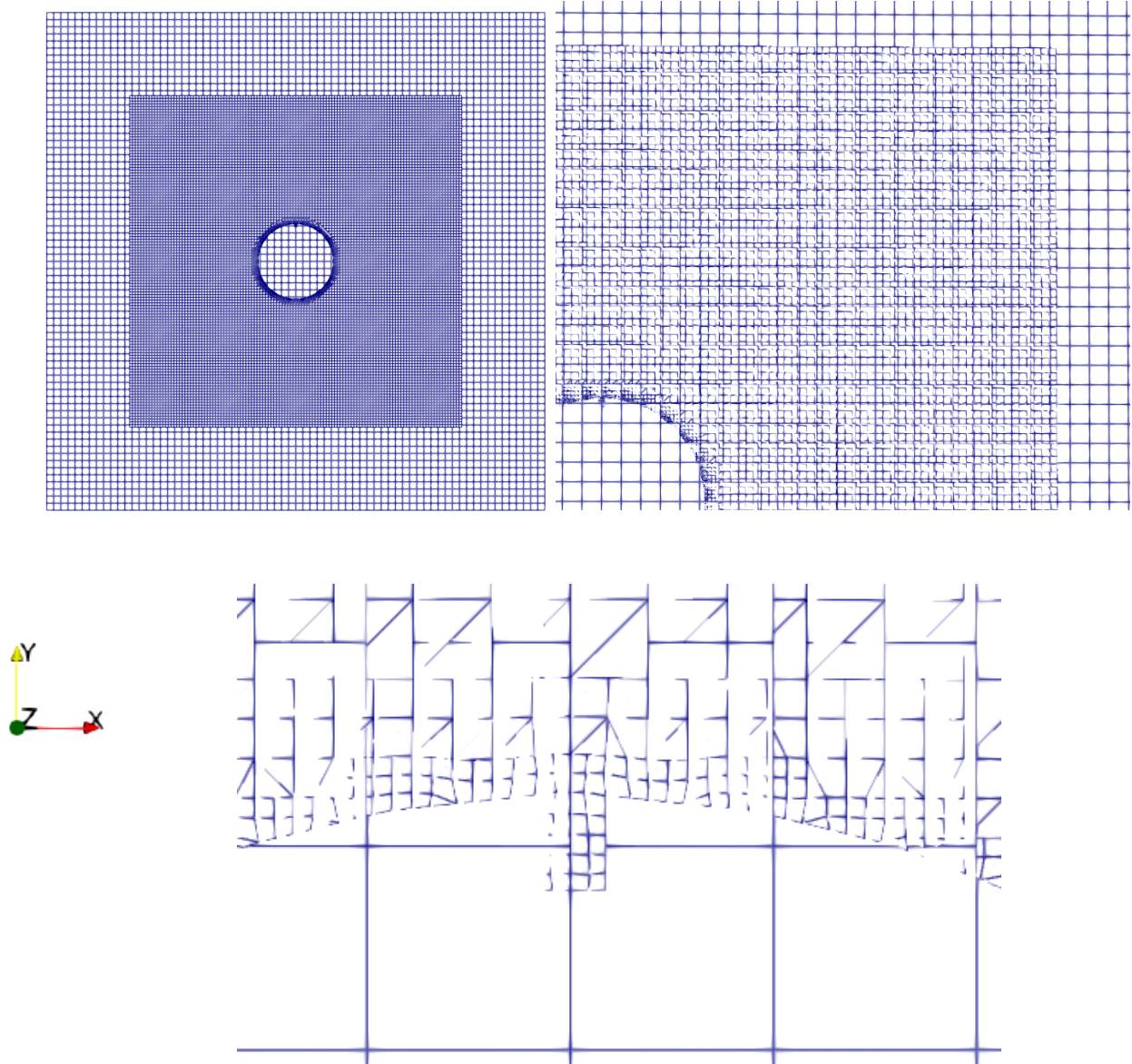


Figure 6: Final computational mesh showing local refinement near the rotor and discharge orifice.

Table 1: Mesh quality statistics obtained using `checkMesh`.

Parameter	Value
Maximum aspect ratio	2.21
Maximum non-orthogonality	25.14°
Average non-orthogonality	2.68°
Maximum skewness	0.87
Maximum cell openness	1.93×10^{-16}
Negative cell volumes	None
Coupled point matching error	0

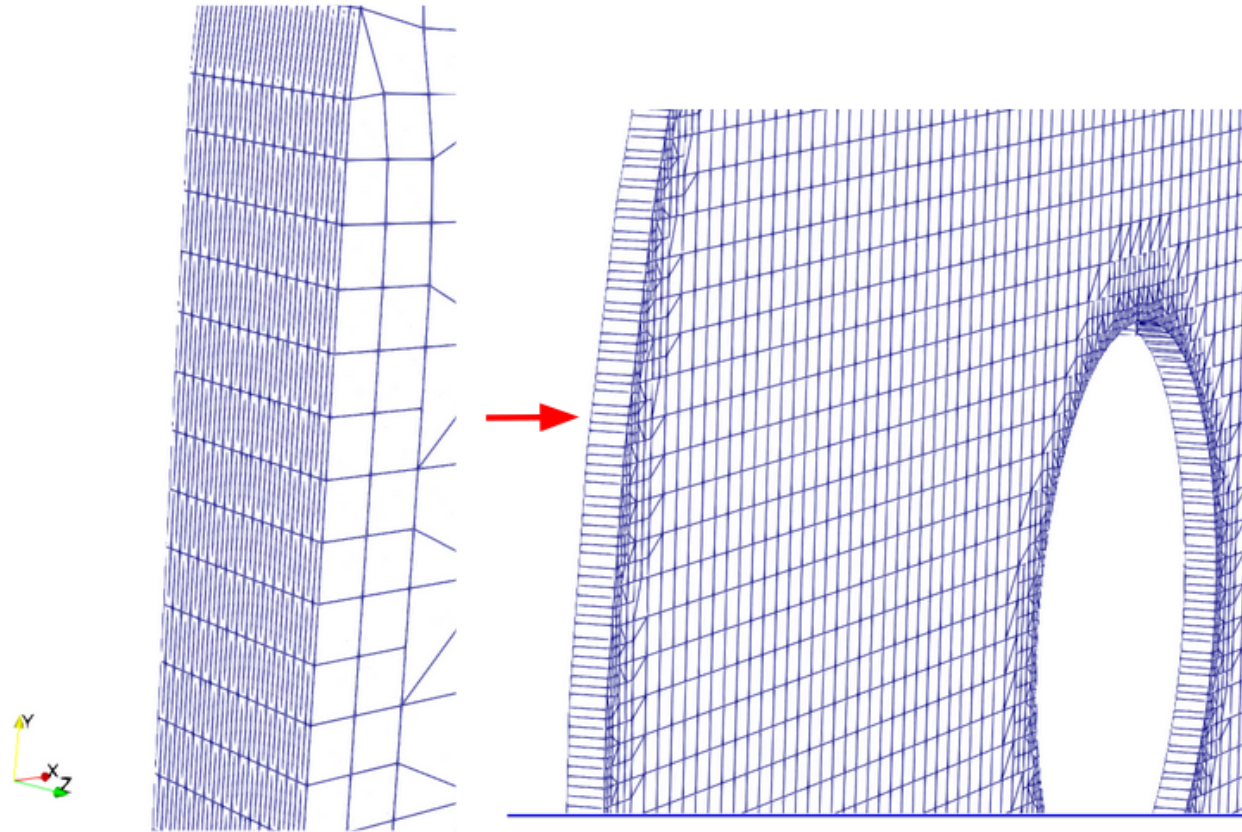


Figure 7: Computational mesh after applying the `flattenMesh` utility, showing the conversion of the three-dimensional mesh into a planar two-dimensional mesh.

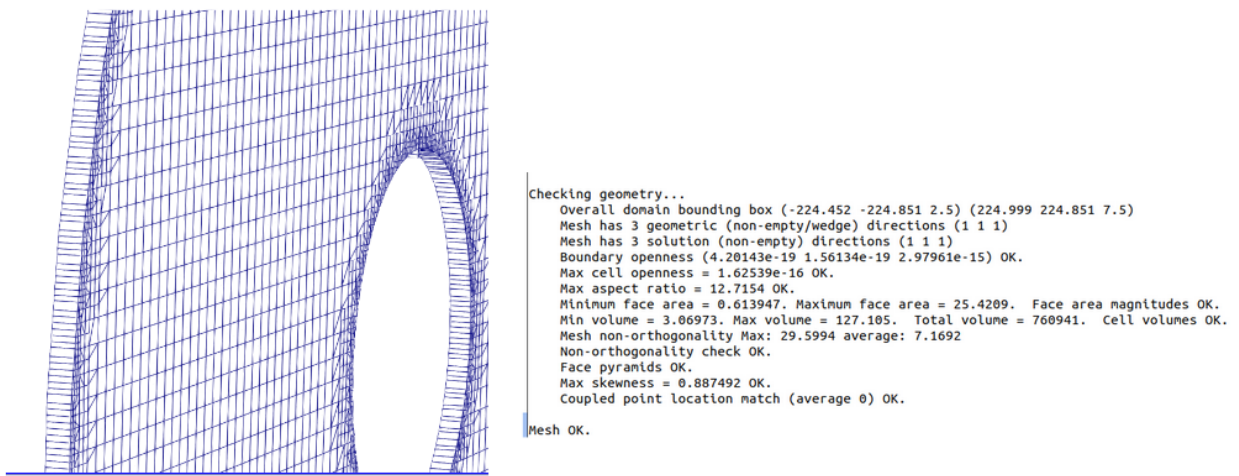


Figure 8: Final two-dimensional computational mesh obtained after applying the `extrudeMesh` utility, consisting of a single-cell thickness in the spanwise direction.

dedicated preprocessing scripts (`Allrun.pre` and `Allclean`) for automated mesh generation and cleanup. The top-level `Allrun` script sequentially executes the preprocessing steps for both mesh

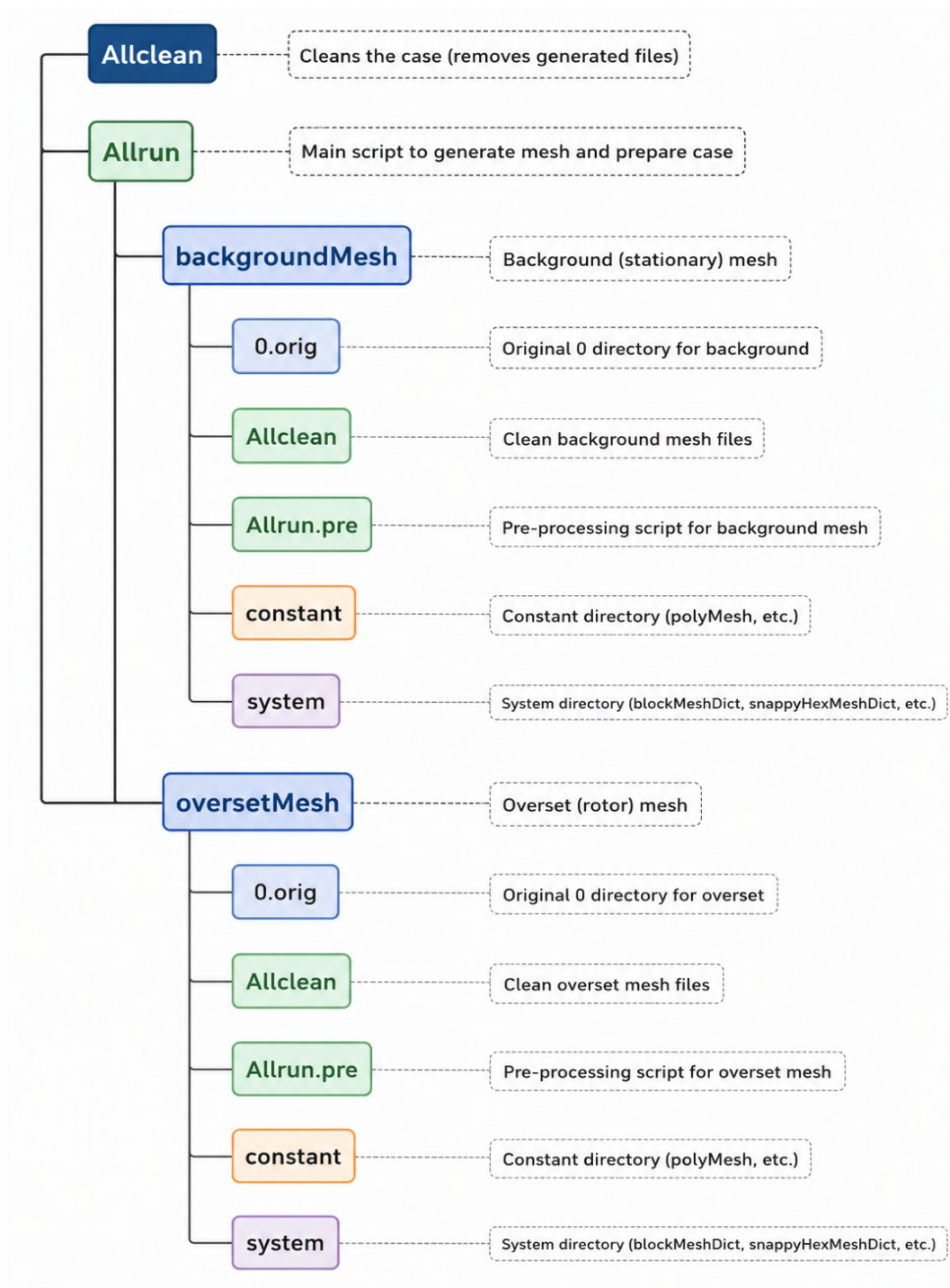


Figure 9: Directory structure of the OpenFOAM case used for the background and overset mesh generation. The case organization includes preprocessing scripts, system dictionaries, and mesh files for both stationary and rotating domains.

regions before combining them into the final computational domain. This modular directory organization improves the reproducibility of the simulation workflow, simplifies mesh management,

and facilitates future modifications to the geometry, mesh resolution, and solver settings.

6 Conclusions

In the present work, an overset mesh framework for a single-orifice rotary slinger atomizer was successfully developed using OpenFOAM. The computational geometry was generated from a CAD model and converted into STL surface files for mesh generation. A systematic preprocessing workflow was established using the OpenFOAM utilities `blockMesh`, `surfaceFeatureExtract`, `snappyHexMesh`, `flattenMesh`, `extrudeMesh`, `mergeMeshes`, and `checkMesh`. The developed workflow enabled the generation of both the stationary background mesh and the rotating overset mesh while maintaining a structured and reproducible case organization.

The generated computational mesh satisfied all OpenFOAM mesh quality requirements. The mesh exhibited acceptable aspect ratio, non-orthogonality, skewness, and cell connectivity, confirming its suitability for overset mesh simulations. The successful integration of the background and rotating meshes demonstrates the capability of the developed framework to model rotating geometries without mesh deformation. Overall, the developed overset mesh methodology provides a robust preprocessing framework that can be extended to various rotating machinery applications and establishes the computational foundation for future multiphase flow simulations of rotary slinger atomizers.

7 Future Work

The present work was limited to the development and verification of the overset mesh framework. Future work will focus on performing transient two-phase flow simulations using the OpenFOAM solver `interDyMFoam`, which combines the Volume of Fluid (VOF) method with dynamic overset mesh capabilities. The developed computational framework will be used to simulate the continuous rotation of the slinger while resolving the liquid-air interface during the atomization process.

The future simulations will investigate the liquid flow inside the rotating slinger, liquid discharge through the single orifice, primary liquid sheet formation, ligament development, and droplet breakup under different rotational speeds and liquid flow rates. The numerical predictions of spray evolution, droplet size distribution, and breakup characteristics will be compared with available experimental observations for validation. Furthermore, the developed framework can be extended to multiple-orifice slinger geometries and more realistic operating conditions, providing a computational platform for studying rotary atomization and optimizing slinger atomizer designs.

References

1. Sahu, S., Chakraborty, A., Maurya, D. (2021). Coriolis-induced liquid breakup and spray evolution in a rotary slinger atomizer: Experiments and analysis. *International Journal of Multiphase Flow*, 135, 103532. <https://doi.org/10.1016/j.ijmultiphaseflow.2020.103532>