



iJRASET

International Journal For Research in
Applied Science and Engineering Technology



INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 14 **Issue:** X **Month of publication:** October 2026

DOI: <https://doi.org/10.22214/ijraset.2026.85409>

www.ijraset.com

Call: ☎ 08813907089

E-mail ID: ijraset@gmail.com

Design and Numerical Simulation of a Microfluidic Mixer Using COMSOL Multiphysics

Pratham Utkarsh Vajanam¹, Sagar Shantkumar Koshti², Mahesh Dashrath Survase³, Vishal Syamgir Giri⁴, V. L. Jagtap⁵
^{1, 2, 3, 4}Student of Karmayogi Institute of Technology, Shelve, Pandharpur, Maharashtra, India

Abstract: Microfluidic devices are used for the handling of small volumes of fluid by means of microscale channels and are generally regarded as suitable for chemical analysis, biomedical testing, pharmaceutical processing, and for use in lab-on-a-chip systems. It is difficult to achieve efficient mixing when the Reynolds number is low since the flow is usually laminar and turbulent eddies are either weak or missing. The present paper describes a numerical-study framework which is used for the design and simulation of a passive microfluidic mixer with the help of COMSOL Multiphysics. The method employed involves a three-dimensional channel geometry, laminar-flow modelling, and the modelling of species transport in order to assess the velocity, pressure, concentration distribution, and the quality of mixing at the outlet. A mixing index based on concentration variance is introduced in order to allow a quantitative comparison between different design cases. The paper also establishes a repeatable procedure for carrying out mesh-independence checks and for conducting parametric studies on inlet velocity, channel dimensions, and the geometry of the mixing element. The reference study by Jomha, Oruç, and Yayla, which used ANSYS Fluent to investigate an optimised zigzag split-and-recombination micromixer, is included here solely as a benchmark in the existing literature and is not presented as a result from the current COMSOL model. The numerical values from the user's model have to be entered after the COMSOL solution and the derived quantities have been verified. For this reason, no mixing-index or pressure-drop value that has not been verified is stated in this draft.

I. INTRODUCTION

Microfluidics is the science and engineering of controlling small quantities of fluids in channels with characteristic dimensions commonly ranging from micrometres to millimetres. Miniaturized fluid handling can reduce sample and reagent consumption, shorten analysis time, and support integration of several laboratory operations on a compact platform. Applications include point-of-care diagnostics, DNA and biochemical analysis, drug formulation, chemical synthesis, and pharmaceutical processing.

Mixing is one of the major challenges in microchannels. At small length scales and low Reynolds numbers, the flow is often laminar, so adjacent streams do not mix rapidly through turbulence. Molecular diffusion is responsible for transport across the interface, and the time required for mixing can be long when the diffusion distance is large. A mixer should therefore create a larger fluid–fluid interface, stretch and fold the interface, or repeatedly split and recombine streams while maintaining an acceptable pressure drop.

Micromixers are commonly grouped into active and passive devices. Active devices use external energy such as electric, magnetic, acoustic, or mechanical actuation. Passive devices rely mainly on channel geometry and the imposed flow. Passive configurations are attractive because they can be relatively simple to fabricate and do not require a separate actuation subsystem. However, performance depends strongly on geometry, flow rate, fluid properties, and the diffusion coefficient of the transported species.

The present project focuses on designing and numerically evaluating a microfluidic mixer using CAD geometry and COMSOL Multiphysics. The objective is to establish a consistent simulation workflow, quantify mixing at the outlet, and identify which geometric or operating parameters should be adjusted to improve mixing without causing an excessive pressure drop.

A. Objectives

- Develop or import a three-dimensional microchannel geometry for a passive micromixer.
- Set up laminar flow and species-transport physics in COMSOL Multiphysics.
- Apply suitable inlet, outlet, wall, and concentration boundary conditions.
- Evaluate velocity and concentration fields and calculate the outlet mixing index.
- Check mesh independence and compare design cases using mixing index and pressure drop.
- Document the model so that its inputs and results can be reproduced.

II. METHODOLOGY

A. Passive split-and-recombination mixing

The paper by Jomha, Oruç and Yayla looks at a passive micromixer which is based on a zigzag split-and-recombination (SAR) idea. The researchers examine Reynolds numbers in the range from 1 to 100 and apply a Taguchi design-of-experiments method in order to compare various geometric factors, such as the zigzag angle, the number of periods, and the dimension ratios. They state that their preferred configuration is one with a 70° angle, four periods, $w/d = 0.125$, and $w/t = 1$. The study assesses mixing by means of concentration variance and also takes the pressure drop into account. These values are characteristics of the design that has been published and should not be regarded as applicable to the current model unless the user's CAD dimensions and conditions are the same.

B. Relevance of numerical simulation

Computational fluid dynamics provides access to velocity, pressure, and concentration distributions that are difficult to measure throughout a microscale channel. A useful numerical study must specify the geometry, fluid properties, inlet flow conditions, outlet pressure, wall conditions, species diffusivity, mesh, solver settings, and convergence criteria. A mesh study is important because concentration gradients can be steep near the interface between two streams and near geometric turns.

C. Research gap addressed by this project

- A specific geometric-based COMSOL protocol is required to ensure that all CAD measurements, physics choices, and boundary conditions are recorded in a consistent manner.
- The visual contours of concentration alone do not constitute sufficient comparison; rather, a numerical value for outlet mixing index should also be included along with pressure drop values.
- Mesh independence and convergence need to be proven before comparing the various geometries.
- Inlet velocity/Reynolds number needs to be isolated from the geometric factor so that any performance difference can be attributed to the correct parameter.
- The existing ANSYS based benchmark needs to be compared against the new COMSOL case once the geometry, fluid parameters, diffusivity, and boundary conditions match.

D. Geometry and Physical Model

Figure 1 is an example of a concept diagram of a two inlet passive micromixer design that consists of a turning section for mixing followed by one outlet.

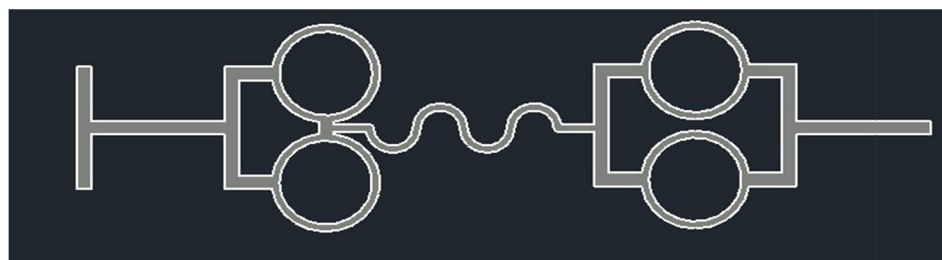


Fig. 1. Conceptual illustration of a passive two-inlet microfluidic mixer.

E. Geometry parameters to report

Parameter	Symbol	value
Channel width/Length	w	1/6.35(mm)
Channel depth	h or t	0.1(mm)
Inlet width	w_in	0.1(mm)
Outlet width	w_out	0.1(mm)
Mixing-section length	L_m	6.35(mm)
Circle dimension outer/inlet	r	0.4/0.35(mm)

Table 1. Geometrical Parameters and Dimensions of the Microfluidic Mixer Channel

F. Numerical Methodology

The problem should be modeled as steady and incompressible with Newtonian flow and species transport for dilute mixtures unless another physical description is required for the application that you wish to solve. It is imperative that the exact physics interfaces and materials properties be obtained from the COMSOL model.

a) Governing equations

Continuity equation for incompressible flow:

$$\nabla \cdot \mathbf{u} = 0$$

Momentum equation for steady laminar flow:

$$\rho (\mathbf{u} \cdot \nabla) \mathbf{u} = -\nabla p + \mu \nabla^2 \mathbf{u}$$

Convection–diffusion equation for a transported species concentration c :

$$\mathbf{u} \cdot \nabla c = \nabla \cdot (D \nabla c)$$

In this case, $\mathbf{u}$ stands for velocity, ρ for density, p for pressure, μ for dynamic viscosity, c for concentration of species, and D for molecular diffusion coefficient. Such equations are derived under the assumption of steady state and uniform properties. In case of transient analysis or multicomponent diffusion approach, these governing equations need to be modified.

Boundary	Suggested condition	Value
Inlet 1	Inlet velocity 1	0.001(m/s)
Inlet 2	Inlet velocity 2	0.001(m/s)
Outlet	Pressure reference, commonly $p = 0$ Pa; convective species outflow	0 Pa
Channel walls	No-slip velocity; zero normal species flux	No slip
Initial values	Concentration 1/Concentration 2	1/0

Table 2. Boundary Conditions and Initial Values for the Microfluidic Mixer Simulation

When two liquids that can mix together are depicted by one liquid with an immiscible species, it is necessary to state how the species concentration depicts the two flows. When the two liquids have different densities or viscosities, one constant property liquid is inadequate for the assumption

b) Mixing index

The mixing index should be computed based on the sampling of the outlet concentration field. One typical expression for the mixing index, using variance, is $MI = 1 - \sigma/\sigma_0$, where σ is the standard deviation of the outlet concentration field and σ_0 is the standard deviation of the corresponding unmixed reference state, both normalized in the same way. The exact definition of the unmixed reference state should be provided. One possibility would be to extract the outlet concentration from the simulation results in COMSOL and compute the mean value and standard deviation based on the area-weighted sampling, where applicable, and normalize with the corresponding unmixed concentration field. Do not use an arithmetic mean of the unequally spaced samples.

c) Mesh independence and solver convergence

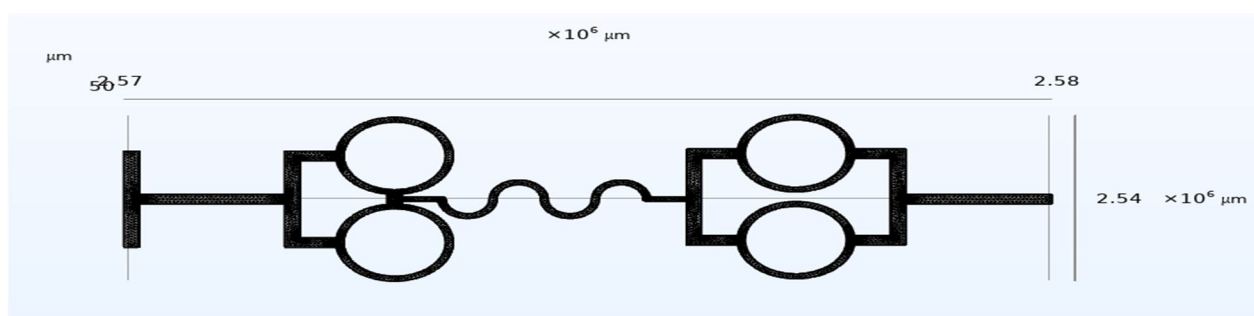


Fig. 2. Geometrical Configuration of the Proposed Micromixer with Circular Chambers and a Wavy Microchannel

Three levels of mesh (coarse, medium, and fine) must be analyzed at minimum. Document the element count, outlet mixing parameter, pressure drop, and relative difference from one level to the next. Mesh should be refined in the vicinity of turns and where the two inlet flows meet. A suitable mesh will produce negligible differences in the relevant quantities once it is refined beyond a certain point. Specify the tolerance and residuals settings used in the solution process.

The figure shows the geometrical configuration of the proposed micromixer designed for microfluidic applications. The design consists of an inlet channel, two circular chambers on the left, a wavy mixing channel at the center, two circular chambers on the right, and an outlet channel.

- Inlet channel: Introduces the working fluids into the micromixer.
- Circular chambers: Provide additional flow regions that may help redistribute the fluid.
- Wavy microchannel: Changes the direction of fluid flow and can promote fluid-interface deformation, improving mixing through convection.
- Outlet channel: Allows the mixed fluid to exit the device.

The proposed geometry is intended to enhance mixing in a compact space. Its performance can be evaluated using COMSOL Multiphysics by studying the mixing index, pressure drop, velocity distribution, and flow behavior.

d) COMSOL Simulation Workflow

- Import or build the correct CAD geometry and verify dimensions and units.
- Confirm continuity of the fluid domain and identification of two inlet and one outlet boundary conditions.
- Import or construct interfaces of Laminar Flow (spf) and Transport of Diluted Species (tds), or other interfaces corresponding to the physical model.
- Provide fluid properties and diffusion coefficient of the species from a documented source or experimentally measured values.
- Apply inlet flow rate/velocity, species concentrations at inlets, outlet pressure, no-slip walls and species fluxes at walls.
- Build the mesh with refinement in the areas of narrow channels, sharp turns and interface area.
- Perform flow simulation and check convergence and velocity-pressure field.
- Perform species transport simulation using the calculated velocity field and observe concentration distribution at the outlet.
- Calculate pressure drop and calculate the outlet mixing index based on the cross-section of the channel documented in literature.
- Repeat the above procedure with at least three different meshes and perform parameter sweep for Reynolds number or geometry parameters.

III. RESULTS

A. Results and Discussion

This part is purposefully designed in accordance with COMSOL outputs of the user. This uploaded MPH file has not been verified separately in this instance for its solved data sets, mesh parameters, boundary conditions, or computed mixing index. Hence, there is no fabrication of numerical data in this instance. Screenshot and numerical data to be imported from the model should be included in the table given below.

B. Velocity and pressure field

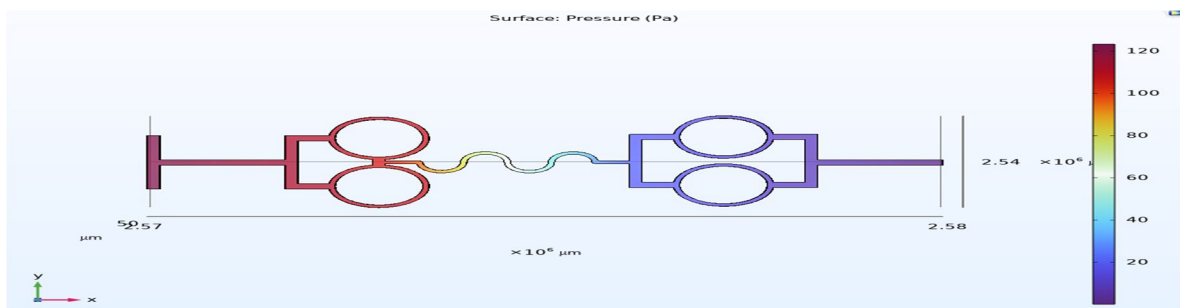


Fig.3. Pressure Distribution in the Proposed Microfluidic Mixer Simulated Using COMSOL Multiphysics

Examine whether the velocity profile is smooth or not, whether there is any recirculation or high-velocity region around any corner, and whether there is uniform reduction in pressure from inlet to outlet. Present the results using a velocity magnitude graph and a pressure profile with proper unit scale. Recirculation must be supported by velocity vectors/ streamlines.

The pressure distribution in the proposed microfluidic mixer was analysed using COMSOL Multiphysics. Figure 2 illustrates the variation in pressure throughout the microchannel, including the inlet, circular chambers, wavy mixing section, and outlet. The colour contour indicates relatively high pressure near the inlet and lower pressure toward the outlet. This pressure difference drives the fluid through the channel network. The wavy mixing section introduces repeated changes in flow direction, which may enhance transverse fluid transport and mixing. However, the bends and branching regions may also increase hydraulic resistance and pressure losses. Therefore, pressure distribution is an important parameter for evaluating the performance of the proposed micromixer. The design should be assessed by considering both the pressure drop and mixing efficiency to achieve effective mixing with minimum pumping requirements.

C. Concentration distribution and mixing

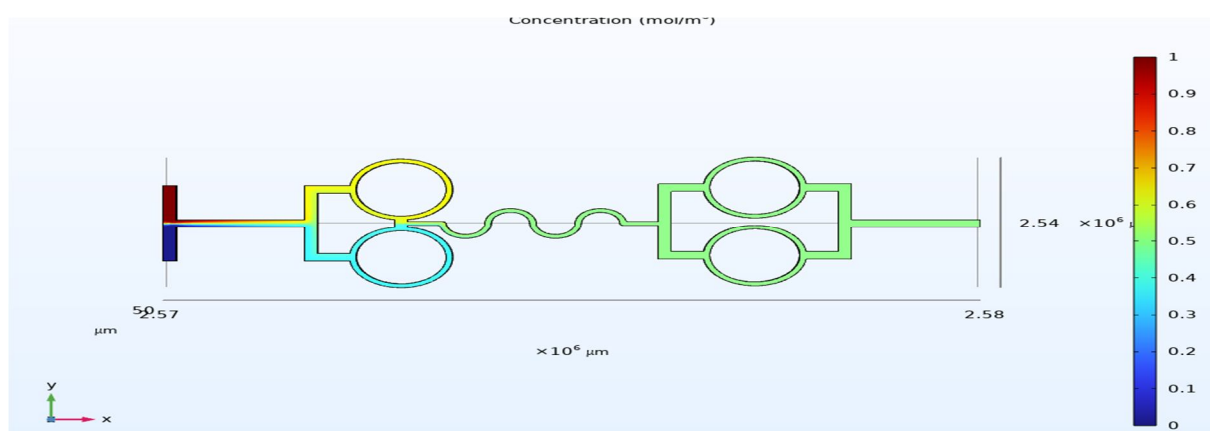


Fig. 4. Concentration Distribution and Mixing Performance in the Proposed Microfluidic Mixer Using COMSOL Multiphysics.

Make use of the contour of concentration at the outlet, together with other cross-sections, if possible, within the mixing region. A properly mixed outlet will have a fairly homogeneous concentration profile instead of two separate concentration bands. The numerical mixing index is to be reported together with its computation methodology.

D. Pressure-drop and mixing trade-off

The pressure distribution in the proposed micromixer was analyzed using COMSOL Multiphysics. The contour plot shows a pressure variation along the microchannel, with higher pressure near the inlet and lower pressure toward the outlet. The curved mixing section and circular chambers influence the flow path and contribute to mixing by redirecting the fluid. However, these features may also increase flow resistance and pressure drop. Therefore, the micromixer design must balance improved mixing efficiency with a reasonable pressure drop to minimize pumping requirements. The actual pressure drop should be calculated by subtracting the outlet pressure from the inlet pressure.

Though geometric turns may lead to higher stretching of the interface and better mixing, they might also result in higher hydraulic resistance. Thus, the evaluation of designs must include both MI and Δp parameters. Not necessarily will be the design with the maximum MI value the most preferable one.

Limitations

- The geometric representation used in this manuscript is purely theoretical and does not form part of any MPH model presented.
- The solutions to the COMSOL models used in this manuscript have not been independently verified.
- It is noted that no experimental verification has been provided; the numerical findings should therefore be considered simulations.
- The manuscript needs to clarify whether the liquids are being modeled as one carrier fluid with a transported species or as two different fluids.
- The normalization method used for the mixing index and area weighting needs to be explained.

IV. CONCLUSION

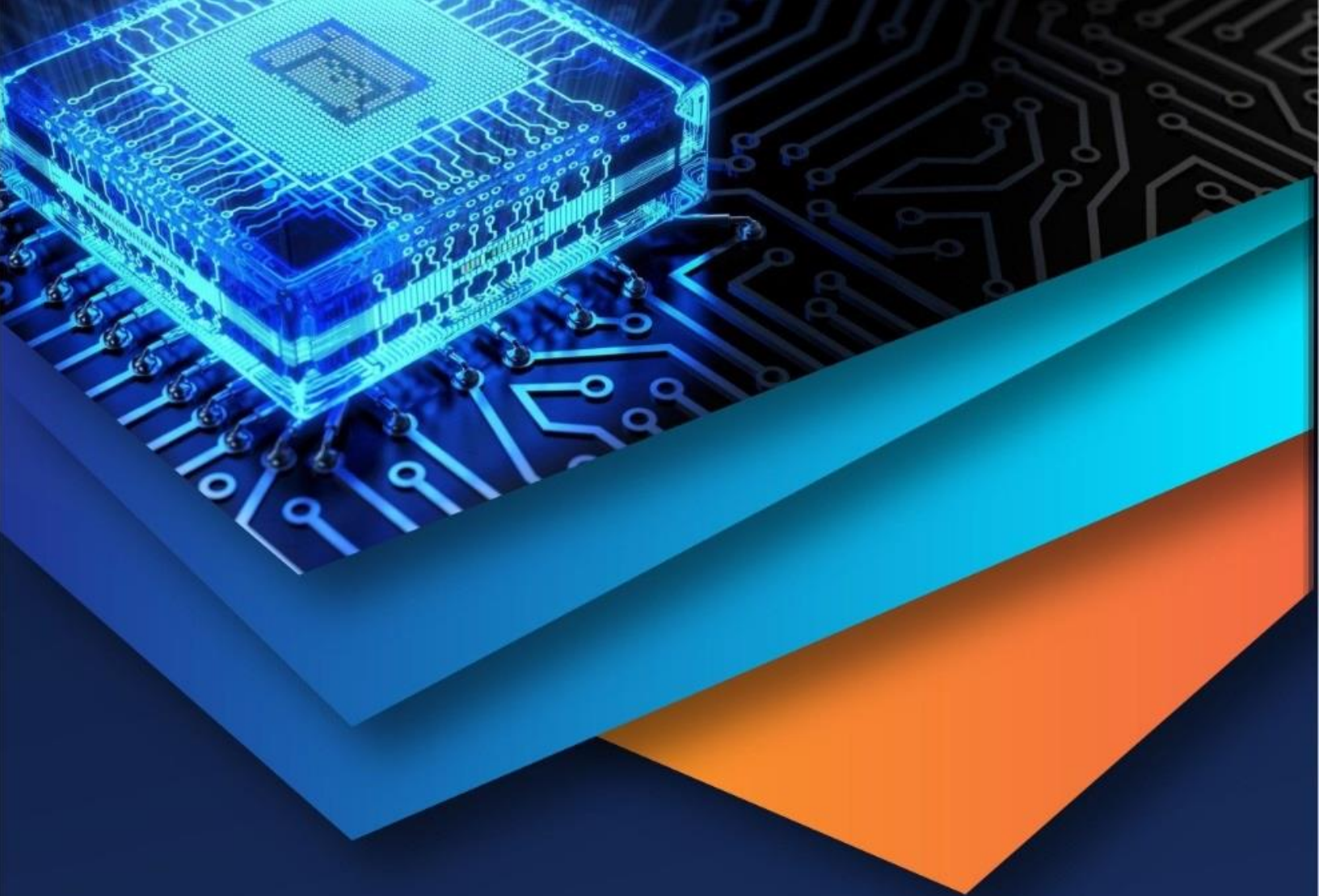
The numerical approach to analyzing a passive microfluidic mixer based on CAD and COMSOL Multiphysics is introduced in this manuscript. The methodology employs both laminar flow equations and species transport equations with boundary conditions explicitly stated. Mesh independence test and outlet concentration variance mixing criteria are also incorporated into the methodology. Previous literature suggests that splitting and recombining and zigzag designs improve interfacial stretching; however, both designs need to be evaluated with respect to pressure drop as well.

Conclusions regarding this particular design will be drawn only after the actual COMSOL outputs have been obtained.

Further work may include a mesh independent parametric analysis, geometry comparisons, optimization of inlet velocity and dimensions of mixing element, experimental work with dyes/concentrations, and fabrication feasibility.

REFERENCES

- [1] M. Jomha, M. Oruç, and S. Yayla, "Numerical investigation of design of a passive micromixer compound shape on split and recombination concept," Pamukkale University Journal of Engineering Sciences, vol. 32, no. 4, pp. 638–645, 2026, doi: 10.65206/pajes.03372.
- [2] Y. Liao, Y. Mechulam, and B. Lassalle-Kaiser, "A millisecond passive micromixer with low flow rate, low sample consumption and easy fabrication," Scientific Reports, vol. 11, Art. no. 20119, 2021, doi: 10.1038/s41598-021-99471-x.
- [3] Q. Fu, Z. Liu, S. Cao, Z. Wang, and G. Liu, "Topology-optimized micromixer design with enhanced reverse flow to increase mixing efficiency," Micromachines, vol. 14, no. 8, Art. no. 1599, 2023, doi: 10.3390/mi14081599.
- [4] B. Yuan, S. Yuan, and H. Wang, "Numerical and experimental investigation of mixing enhancement in a zigzag passive micromixer with D-shaped obstacles," Micromachines, vol. 17, no. 2, Art. no. 190, 2026, doi: 10.3390/mi17020190.
- [5] A. Kolesnik et al., "A plug-and-play volume minimizing micromixer," Advanced Science, 2026, doi: 10.1002/advs.202510268.
- [6] A. A. Nizar and I. S. L. Abdul Hamid, "Design and simulation of passive micromixer in microfluidic device," Evolution in Electrical and Electronic Engineering, vol. 2, no. 2, pp. 102–109, 2021.
- [7] V. Hessel, H. Löwe, and F. Schönfeld, "Micromixers—a review on passive and active mixing principles," Chemical Engineering Science, vol. 60, nos. 8–9, pp. 2479–2501, 2005, doi: 10.1016/j.ces.2004.11.033.
- [8] A. D. Stroock, S. K. W. Dertinger, A. Ajdari, I. Mezić, H. A. Stone, and G. M. Whitesides, "Chaotic mixer for microchannels," Science, vol. 295, no. 5555, pp. 647–651, 2002, doi: 10.1126/science.1066238.
- [9] A. D. Stroock, S. K. W. Dertinger, G. M. Whitesides, and A. Ajdari, "Patterning flows using grooved surfaces," Analytical Chemistry, vol. 74, no. 20, pp. 5306–5312, 2002, doi: 10.1021/ac0257389.
- [10] F. Pennella et al., "Numerical and experimental characterization of a novel modular passive micromixer," Biomedical Microdevices, vol. 14, no. 5, pp. 849–862, 2012, doi: 10.1007/s10544-012-9665-4.



10.22214/IJRASET



45.98



IMPACT FACTOR:
7.129



IMPACT FACTOR:
7.429



INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Call : 08813907089  (24*7 Support on Whatsapp)