FOSSEE, IIT Bombay
OpenFOAM Case Study Project
May 25, 2026

FSI of Rotating Porous Square Cylinder Using Continuous Forcing Immersed Boundary Method

Prajwol Bohara[1],Manjil Sitoula[2],Anirudha Kulkarni[3]

[1]Department of Automobile and Mechanical Engineering, IOE Thapathali Campus, Institute of Engineering, Tribhuvan University, Nepal
[2]Research Assistant,FOSSEE, IIT Bombay
[3]Assistant Professor, Department of Mechanical Engineering (SOT), Pandit Deendayal Energy University, Gandhinagar, GJ

Abstract

This case study presents a comprehensive numerical study of flow past a rotating porous square cylinder using the continuous forcing immersed boundary method. A finite volume approach combined with the immersed boundary method (IBM) and moving boundary treatment is employed to obtain the numerical results. The investigation covers steady flow regimes ($10 \leq Re \leq 40$) with different values of permeability (Darcy number 10^{-6} to 10^{-3}) and porosity values between $\varepsilon = 0.148$ and 0.887. Numerical simulations are carried out using a modified version of the `pimpleFoam` solver, while computational background cartesian mesh is generated using the OpenFOAM `blockMesh` utility. The case study aims to understand the influence of the Darcy number on drag coefficient characteristics and the underlying flow physics for stationary and rotating porous bodies. In the current study, only a one-way coupled fluid–structure interaction (FSI) approach is considered.The geometric domain, parameters and flow conditions are summarized in Table 1 and Fig. 1

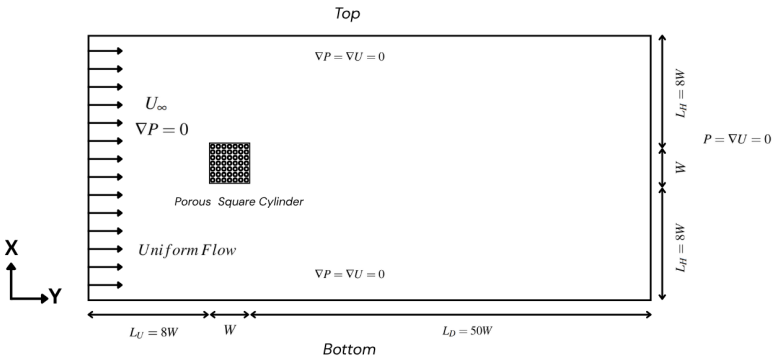

Figure 1: Geometry and domain dimensions

Parameter	Symbol	Value (m)
Upstream length	L_u	4
Square porous obstacle width	W	0.5
Downstream length	L_d	25.0
Height	H	8.5

Table 1: Geometry definitions of the background Cartesian mesh as in Fig. 1.