Implementing custom boundary conditions using 'codedFixedValue' boundary condition in OpenFOAM

Mohamed Adnan K N

Working Professional

ABSTRACT

In Computational Fluid Dynamics, boundary conditions play a crucial role in defining the behavior of physical systems. Among these, the fixed value boundary condition is widely used for its simplicity and effectiveness in various applications. 'codedFixedValue' boundary condition in OpenFOAM is a powerful technique that integrates custom C++ coding to expand the versatility and implementation of traditional fixed value boundary conditions. This report explores the application of the 'CodedFixedValue' Boundary Condition, a method that integrates coding techniques to enhance the implementation and versatility of fixed value boundary conditions. The study focuses on a 2D laminar mixed convection heat transfer problem within a lid-driven cavity as a base case. This report outlines the step-by-step process of implementing this custom boundary condition, including coding, validation, and testing within the OpenFOAM environment. The geometry and mesh are generated using the 'blockMesh' utility. The unsteady, laminar flow is simulated using the 'buoyantPimpleFoam' solver, which is well-suited for solving buoyancy-driven flows in both transient and steady-state conditions. Details regarding geometry and flow has been listed in Table-1.

Table 1: Details of geometry and flow conditions

Geometry details	Length of the cavity(x) = 1m Height of the cavity (y) = 1m Depth of the cavity (z) = 1m
Fluid Property	Prandtl number (Pr) = 0.71 Grashof number (Gr_T) = 100 Dynamic Viscosity (μ)= 0.0966 N.s/m^2
Reynolds number	Re = 100