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

Mohamed Adnan K N

Working Professional

ABSTRACT

This report explores the implementation and application of the codedFixedValue boundary condition in OpenFOAM. 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. The report demonstrates various applications of codedFixedValue through examples, such as time-varying, space-varying, output-based input and condition-based boundary conditions in a two-dimensional lid-driven cavity flow with mixed convection. The buoyancy-driven flow is modeled using the buoyantPimpleFoam (Boussinesq's approximation) solver. Geometry and mesh have been generated by using 'blockMesh' utility. 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
Reynolds number	Re = 100
Thermal expansion coefficient	$\beta_{T,air} = 0.00317\ K^{-1}$
Prandtl number	Pr = 0.71
Grashof number	$Gr_T = 100$