

عنوان مقاله:

A Novel Momentum-Variable Based Procedure Development via OpenFOAM to Solve Turbulent Natural Convection in Enclosures

محل انتشار:

هفدهمین کنگره ملی مهندسی شیمی ایران (سال: 1400)

تعداد صفحات اصل مقاله: 6

نویسندگان:

Farshad Rahimi - *Center for Process Design, Safety and Loss Prevention (CPSL), Department of Chemical and Petroleum Engineering, Sharif University of Technology, Tehran, Iran*

Davood Rashtchian - *Center for Process Design, Safety and Loss Prevention (CPSL), Department of Chemical and Petroleum Engineering, Sharif University of Technology, Tehran, Iran*

Masoud Darbandi - *Department of Aerospace Engineering, Sharif University of Technology, Tehran, Iran*

خلاصه مقاله:

Natural buoyant convection flow is an exemplary heat transfer flow with lots of increasing applications in industries. So, it is very crucial to model this convection flow accurately. This reserach focuses on developing a procedure to model buoyant driven turbulent flows. To validate the developed procedure, the current results are compared with experimental data collected in a square cavity ($Ra=1.58 \times 10^9$). Considering the geometry and condition of the test case, the flow is turbulent. The momentum-variable based formulation procedure is proposed due to its advantages compared with velocity-based methods. OpenFOAM ۴.۱ is selected as a platform to develop this procedure. Modeling has been done for different mesh sizes and turbulence models. Nusselt number is selected as an indicator to ensure mesh size independency, and proper mesh size is found for modeling. The Comparison between modeling and experimental results shows that the developed procedure is capable of modeling thermos-buoyant turbulent flows with very good accuracy.

کلمات کلیدی:

Natural Convection; Buoyant-driven flow; CFD; OpenFOAM; Turbulent flow

لینک ثابت مقاله در پایگاه سیویلیکا:

<https://civilica.com/doc/1378192>

