FOSSEE, IIT Bombay
OpenFOAM Research Migration Project
April 22, 2022

CFD analysis of fluid flow and heat transfer in a shell and tube heat exchange in OpenFOAM.

Aabhushan Regmi
FOSEE Team, IIT Bombay

Synopsis

In this study chtMutliRegionSimpleFoam is used for the numerical simulation of a shell and tube type heat exchanger. The structure of such type of heat exchanger consists of large pressure vessel inside which is a bundle of tubes carrying fluid, heat transfer takes place through the surfaces of these smaller tubes. This is a case of conjugate heat transfer in multi-region (between two fluids through a solid in between). Turbulent and steady state study is done. OpenFOAM-v2012 is used for this study. The profiles of velocity and temperature at the end of the flow is obtained from the simulation and validated against the value from a published paper.

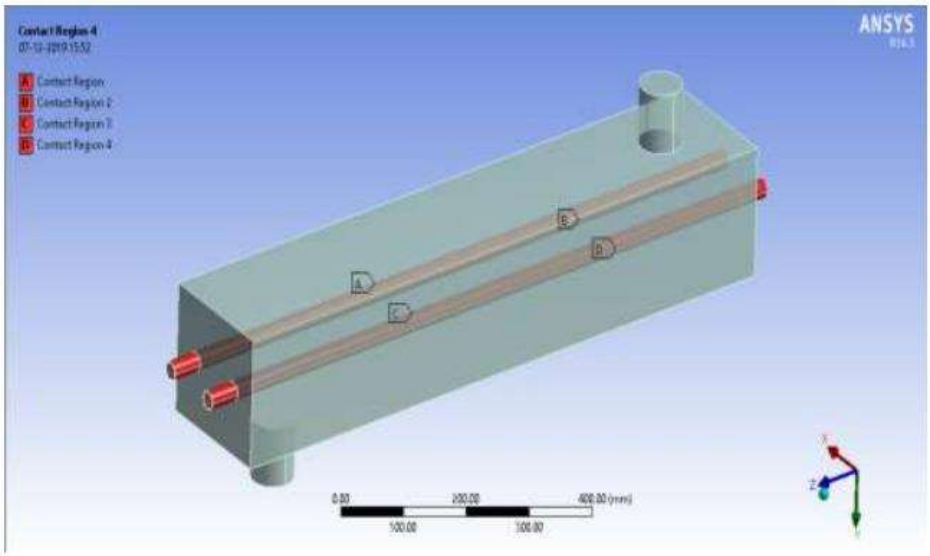

Figure 1: Geometry being Considered

References

CFD ANALYSIS OF HEAT EXCHANGER MODELS DESIGN USING ANSYS FLUENT, Ram Kishan, Devendra Singh, Ajay Kumar Sharma*