TURBULENCE MODELS FOR NUMERICAL SIMULATION OF TEMPERATURE DISTRIBUTION IN SCWR

Parimala Rangan Fidel Castro
Scientific supervisor: Korotkikh A.G., Doctor of Physics and Mathematical Sciences
Tomsk Polytechnic University, 30 Lenina Avenue, Tomsk 634050 Russian Federation
E-mail: fidelcastroprp@gmail.com

In the present study, Computational fluid dynamics (CFD) simulation was conducted for 2×2 rod bare bundle using water at supercritical pressures. Main objective of the simulation is to compare calculation results with varying temperatures. CFD simulation was performed to replicate the results from the experiment of heat transfer to supercritical water in 2 × 2 rod bundle conducted at Shanghai Jiao Tong University [1]. This report presents the results to assess capability of the commercial CFD software Ansys fluent in simulating the convective heat transfer of water at supercritical pressures in nuclear fuel rod. The type of flow for simulation is taken as steady state flow. The mass flux is 800 kg/m2s and the heat flux is 600 kW/m2. The experiment was performed for the pressure of 25MPa. The temperature varies from 300°C, 340°C and 380°C. This simulation is conducted for steady state i.e. all the physical properties of water such as density and viscosity are considered as constant K-epsilon turbulence model is used for our CFD simulation.

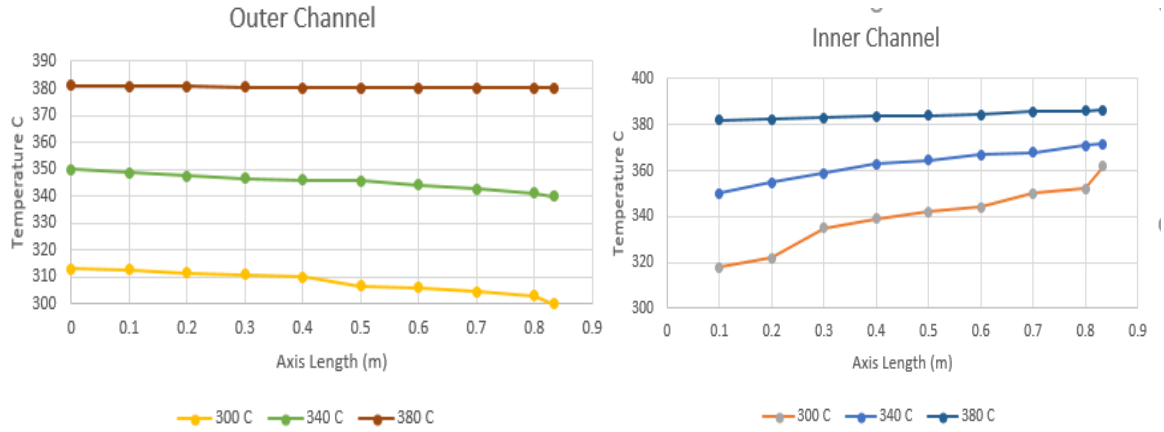

Figure 1 - Dependence of the coolant temperature at 300°C, 340°C, 380°C on the channel length

The results obtained from CFD simulation comes in close agreement with the experimental data as shown in fig1. Temperature is plotted along the radial length for inner and outer channels. The graph obtained is compared with the experimental results. The results were obtained for three inlet temperature of 300 °C, 340°C and 380 °C for 25 MPa pressure.

1. H.Y. Gua, Z.X. HuD. Liua; X. Cheng, Experimental studies on heat transfer to supercritical water in 2 × 2 rod bundle with two channels //Nuclear Engineering and Design Volume 291, September 2015, Pages 212-223.