Computational Fluid Dynamics Simulation of Turbulent Convective Heat Transfer in Rectangular Mini-Channels for Rocket Cooling Applications

Authors : O. Anwar Beg, Armghan Zubair, Sireetorn Kuharat, Meisam Babaie

Abstract : In this work, motivated by rocket channel cooling applications, we describe recent CFD simulations of turbulent convective heat transfer in mini-channels at different aspect ratios. ANSYS FLUENT software has been employed with a mean average error of 5.97% relative to Forrest's MIT cooling channel study (2014) at a Reynolds number of 50,443 with a Prandtl number of 3.01. This suggests that the simulation model created for turbulent flow was suitable to set as a foundation for the study of different aspect ratios in the channel. Multiple aspect ratios were also considered to understand the influence of high aspect ratios to analyse the best performing cooling channel, which was determined to be the highest aspect ratio channels. Hence, the approximate 28:1 aspect ratio provided the best characteristics to ensure effective cooling. A mesh convergence study was performed to assess the optimum mesh density to collect accurate results. Hence, for this study an element size of 0.05mm was used to generate 579,120 for proper turbulent flow simulation. Deploying a greater bias factor would increase the mesh density to the furthest edges of the channel which would prove to be useful if the focus of the study was just on a single side of the wall. Since a bulk temperature is involved with the calculations, it is essential to ensure a suitable bias factor is used to ensure the reliability of the results. Hence, in this study we have opted to use a bias factor of 5 to allow greater mesh density at both edges of the channel. However, the limitations on mesh density and hardware have curtailed the sophistication achievable for the turbulence characteristics. Also only linear rectangular channels were considered, i.e. curvature was ignored. Furthermore, we only considered conventional water coolant. From this CFD study the variation of aspect ratio provided a deeper appreciation of the effect of small to high aspect ratios with regard to cooling channels. Hence, when considering an application for the channel, the geometry of the aspect ratio must play a crucial role in optimizing cooling performance.

Keywords : rocket channel cooling, ANSYS FLUENT CFD, turbulence, convection heat transfer
Conference Title : ICHTFM 2018 : International Conference on Heat Transfer and Fluid Mechanics
Conference Location : Dublin, Ireland
Conference Dates : August 16-17, 2018