Numerical Study of Heat Transfer in Square Duct with Turbulators

Authors : M. H. Alhajeri, Hamad M. Alhajeri, A. H. Alenezi

Abstract : Computational fluid dynamics (CFD) investigation of heat transfer in U-duct with turbulators is presented in this paper. The duct passages used to cool internally the blades in gas turbine. The study is focused in the flow behavior and the Nusselt number (Nu) distributions. The model of the u-duct contains two square legs that are connected by 180* turn. Four turbulators are located in each surface of the leg and distributed in a staggered arrangement. The turbulator height and width are equal to 0.1 of the duct width, and the turbulator height is 0.1 of the distance between the turbulators. The Reynolds number (Re) used in this study is 95000 and the inlet velocity is 10 m/s. It was noticed that, after the flow resettles from the interruptions generated by the first turbulator or the turn, the flow construct two eddies, one large and the other is small after and before the turbulator, respectively. The maximum values of the Nu are found at a distance of approximately one turbulator width w before of the flow reattachment point.

Keywords : computational fluid dynamics, CFD, rib, heat transfer, blade

Conference Title : ICAMAME 2019 : International Conference on Aerospace, Mechanical, Automotive and Materials Engineering

Conference Location : London, United Kingdom

Conference Dates : October 23-24, 2019