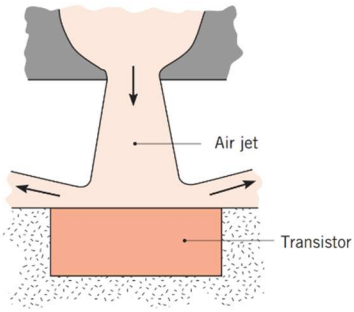
CFD 2D analysis on air jet impingement using chtMultiRegionFoam in OpenFOAM

Jeetkumar Pravinbhai Patel

Department of Mechanical Engineering, Indian Institute of Technology Bombay

Synopsis:

Heat is major issue in any electronic equipment as the temperature of equipment increase there will be lost in operation of it. The heat flux in these devices is in order of 10^5 W/m^2. It is necessary to cool otherwise it is going to fail. Thus, it should be operated in standard temperature range. Among all the cooling technique impingement of fluid is most effective as there is large stagnation zone at contact of impingement. The main objective of this study is to do a simulation of air jet impingement to cool any electronic equipment. Impingement of fluid on any surface has more heat transfer due to large stagnation zone. But what should be the distance between jet exit and heated surface, jet diameter, velocity and velocity profile is required for cooling is parameter of interest. Air jet impingement is studied and has been validated through Martin *at el.* experimental results. This study is on a circular transistor of 10-mm diameter is cooled by impingement of an air exiting a 2-mm-diameter round nozzle with a velocity of 20 m/s and a temperature of 15^{o}C.The jet exit and the exposed surface of the transistor is separated by a distance of 10 mm. If the transistor is well insulated at all but its exposed surface temperature is not to exceed 85^{0}C , what is the heat transfer coefficient?



References

Martin, H., “Heat and Mass Transfer between Impinging Gas Jets and Solid Surfaces,” in J. P. Hartnett and T. F. Irvine, Jr., Eds., Advances in Heat Transfer, Vol. 13, Academic Press, New York, 1977.