

Conjugate Heat Transfer Analysis of a Combustion Chamber using ANSYS Computational Fluid Dynamics to Estimate the Thermocouple Positioning in a Chamber Wall

Authors : Muzna Tariq, Ihtzaz Qamar

Abstract : In most engineering cases, the working temperatures inside a combustion chamber are high enough that they lie beyond the operational range of thermocouples. Furthermore, design and manufacturing limitations restrict the use of internal thermocouples in many applications. Heat transfer inside a combustion chamber is caused due to interaction of the post-combustion hot fluid with the chamber wall. Heat transfer that involves an interaction between the fluid and solid is categorized as Conjugate Heat Transfer (CHT). Therefore, to satisfy the needs of CHT, CHT Analysis is performed by using ANSYS CFD tool to estimate theoretically precise thermocouple positions at the combustion chamber wall where excessive temperatures (beyond thermocouple range) can be avoided. In accordance with these Computational Fluid Dynamics (CFD) results, a combustion chamber is designed, and a prototype is manufactured with multiple thermocouple ports positioned at the specified distances so that the temperature of hot gases can be measured on the chamber wall where the temperatures do not exceed the thermocouple working range.

Keywords : computational fluid dynamics, conduction, conjugate heat transfer, convection, fluid flow, thermocouples

Conference Title : ICFMAE 2020 : International Conference on Fluid Mechanics and Aerospace Engineering

Conference Location : Dublin, Ireland

Conference Dates : December 21-22, 2020