UNSTEADY ANALYSIS OF TURBINE MAIN FLOW COUPLED WITH SECONDARY AIR FLOW

Chunill Hah
NASA Lewis Research Center
Cleveland, Ohio

Two numerical approaches are used to model the interaction between the turbine main gas flow and the wheelspace cavity seal flow. The three-dimensional, unsteady Reynolds-averaged Navier-Stokes equations are solved with a CFD code based on a structured grid to study the interaction between the turbine main gas flow and the wheelspace cavity seal flow. A CFD code based on an unstructured grid is used to solve detailed flow feature in the cavity seal which has a complex geometry. The numerical results confirm various observations from earlier experimental studies under similar flow conditions. When the flow rate through the rim cavity seal is increased, the ingestion of the main turbine flow into the rim seal area decreases drastically. However, a small amount of main gas flow is ingested to the rim seal area even with very high level of seal flow rate. This is due to the complex nature of three-dimensional, unsteady flow interaction near the hub of the turbine stage.