DHIRGHAM A. H. ALKHAFAGIY

"AN AERODYNAMIC STUDY OF DIFFUSER COMBUSTOR FLOW INTERACTION"

SUPERVISORS: Dr. ABDUR RAHIM & Dr. PRABAL TALUKDAR
DEPARTMENT OF MECHANICAL ENGINEERING

ABSTACT

In the gas turbine combustion system, the interaction between the diffuser and combustor external flows plays a key role in controlling the pressure loss, air flow distribution around the combustor liner, and the attendant effects on performance, durability, and stability. The diffuser-combustor flow interaction is governed by several geometric and flow parameters. For a dump diffuser system, these include the pre-diffuser length and area ratio; the dump gap and expansion ratio; the airflow splits between the dome and inner and outer annuli; and the velocity distribution and turbulence level at the compressor exit. Additional considerations are the location of support struts, locating pins, and fuel nozzles. These subcomponents should be designed and placed such that they have minimal effect on the pressure recovery and burner exit temperature quality. The diffuser system should be designed such that there are no unnecessary pressure losses. Further, it should lead to uniform air feed to various liner orifices and cooling devices on the combustor walls. The non-uniformities in the pressure distribution can result in an inadequate cooling film at various locations or can even cause reversal of the flow direction with consequent overheating of the combustor liner. A diffuser design procedure requires as input the various losses in the system. These include the pre-diffuser momentum losses, flow dumping losses, and parametric drag losses of struts, mounts, and fuel nozzles. Currently, these losses are calculated using empirical correlations based on existing experimental data. Such an approach cannot be used effectively for new design

concepts. Further, due to complexity of the geometry and resulting flows, the choice of the optimum configuration depends upon the selection of a large number of parameters. Evidently, the conventional approach of identifying an optimum combination of a large number of parameters through scale model testing followed by selective full scale testing is both expensive and time consuming. These difficulties have led to the development of multidimensional computational methods for analyzing the diffuser and combustor external flows. The flow in a dump diffuser system is comprised of regions of recirculating flows, strong streamline curvature, adverse pressure gradient, developing boundary layer, and impingement flow. Such a flow structure coupled with complex geometric configuration makes the computations of combustor-diffuser flows difficult. Preliminary investigations are carried out for the purpose of validation of CFD code, FLUENT 6.3 with published experimental result. Calibrated results show better agreement with experimental results for both 2-D and 3-D models. An isothermal flow study in the diffuser combustor models has been carried out to investigate the flow behaviors in the annular and can combustors. Annular combustor models investigations involved 2-D simulation study was carried out to evaluate the dynamic parameters and 3-D simulation study was emphases the cooling film and cross flow through the cooling holes, where primary, secondary and dilution holes have high aspects in annulus region. Can combustor studies are carried out to investigate dynamics parameters of the external flow at seven models with different geometry with 2-D simulations. The results show the possibility to improve the external flow performance in dump diffuser-combustor system by new design.