GAS TURBINE ENGINE COMPRESSOR-COMBUSTOR DYNAMICS SIMULATION DESIGN

Sanford Fleeter, Elias Houstis, John Rice and Chenn Zhou

February 16, 1999

INTRODUCTION

The superior output power-to-weight ratio of the gas turbine engine has made it the mainstay of modern aviation propulsion for both civilian and military applications. The capability of producing high output power at low engine weight has also made it a common propulsion system in both marine and armored military ground applications. In addition, gas turbine engines are used for power generation as either stand-alone systems or coupled with steam turbines to power electrical generators and also to power rotating machinery such as oil pumps in large pipeline systems.

Because of the wide uses and high demand for gas turbine engines, several companies produce such engines. As a result, there is fierce global competition among gas turbine manufacturers. To gain an edge in the marketplace, companies are continually striving to improve the efficiency and performance of their engines. This is accomplished through the development and implementation of advanced fan, compressor, combustor and turbine components.

Ideally, this would involve a complete engine simulation, providing the capability for the flow fields through the various components to interact. Thus, the actual physical processes would be simulated, including the effects of three-dimensional, unsteady, turbulent viscous reacting flows and their interaction with the engine structural components.

Unfortunately, a complete engine simulation model requires vast computational resources. As a result, simplified models have been developed that primarily focus on the simulation of steady flow phenomena in the individual engine components. Thus, individual steady flow performance models have been developed for each component. The result is that the performance of the combustor and the turbomachinery are predicted independently. The aerodynamic performance and durability of the fan, compressor and turbine blading are predicted independently of both one another and also independently of the combustor, with the combustor performance predicted independently of the upstream compressor.

Gas Turbine Engine Steady Performance

The surge and choke lines bound the operating range of a gas turbine engine on the compressor aerodynamic performance map, Figure 1.

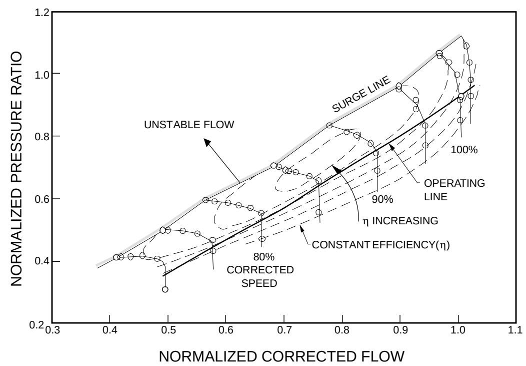

Figure 1. Typical performance map

To assure compressor stability during engine operation, an engine is designed with a surge margin. This entails assuring that the operating point remains a specified distance from the surge line on the performance map. Large surge margins are employed due to transient conditions that move the compressor operating point closer to the surge line. However, a large surge margin that places the compressor operating line far from the surge line can preclude operation at the peak pressure rise or maximum efficiency region. Also, the increase in operational range results in additional flexibility for matching the compressor with the other gas turbine engine components.

The term surge line is actually a misnomer as two types of instability can develop: surge or rotating stall. Surge is a global axisymmetric oscillation of the flow through the compressor, Figure 2, that can include reverse flow during a portion of the surge cycle.

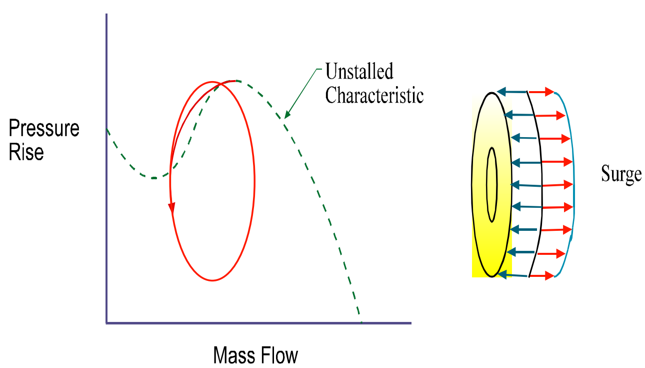

Figure 2. Surge schematic

These oscillations can result in severe damage to the mechanical components of the engine from the unsteady thrust load or the ingestion of combustion gases into the compressor and engine inlet. In a severe surge cycle, the reversed flow through the compressor can extinguish combustion, resulting in a "flame out" or total loss of engine power.

Rotating stall is a local flow deficit that rotates around the compressor annulus, Figure 3. This flow deficit, or cell, is a region in which the local mass flow is near zero.

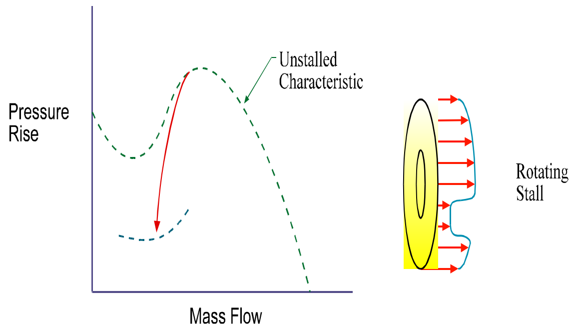

Figure 3. Rotating stall schematic

The rotating stall may consist of one or multiple cells that rotate around the compressor at an angular speed which is a fraction of the rotor speed. This instability results in a loss of compressor performance that may require the shut down of the engine to clear. Operating a compressor in rotating stall can contribute to fatigue damage of the blading resulting from the rotating stall unsteady aerodynamic loading. Also, the loss in compressor performance due to rotating stall can also move the compressor to an operating point where surge is initiated.

Transient Performance

The performance of a gas turbine engine can differ significantly from that predicted from such independent steady flow models. This is because of the inherent unsteady interactions that occur between the various components. The consequences of these dynamics can be quite dramatic, including the unexpected crossing of the compressor surge line while transitioning between engine operating points, as depicted in Figure 4.

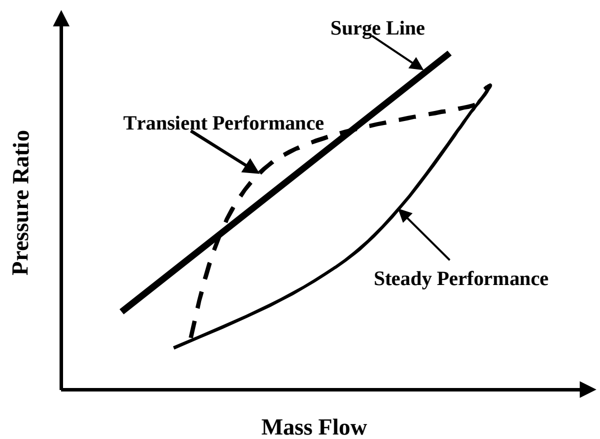

Figure 4. Transient performance

The unexpected crossing of the compressor surge line during engine transients results in a complex dynamic interaction between the engine components driven by rotating stall and surge. These unsteady operating cycles are of particular interest as they result in substantially reduced performance and durability.

This unsteady engine operation produces extreme loading for the turbomachinery blading, resulting in high cycle fatigue (HCF) failures, with surge and rotating stall resulting in dangerous flow induced blade vibrations due to the rapid loading and unloading of the blading. Thus, the accurate analysis of the peak blade row unsteady aerodynamic loading is needed to confidently reduce structural safety factors and associated propulsion system weight.

Recovery from rotating stall and surge is also an important issue since it is impossible to guarantee that an engine can avoid such behavior during its operational lifetime. As a result, it is necessary to accurately simulate the performance of a compressor undergoing a surge transient.

To address the various issues associated with the transient performance of a gas turbine engine, it is thus necessary to implement an advanced simulation that models the entire engine. Thus, this simulation will capture the important compressor-combustor interactions that occur during engine transients, including rotating stall and surge, and their affect on blade row durability.

RESEARCH OBJECTIVE

To address the various issues associated with the transient performance of a gas turbine engine, an advanced simulation will be implemented that models the inlet, fan, compressor and combustor. Thus, this simulation will capture the important compressor-combustor interactions that occur during engine transients, including rotating stall and surge, and their affect on blade row durability. This simulation will provide the capability for the flow fields through the various components to interact. Thus, the actual physical processes will be simulated, including the effects of three-dimensional, unsteady, turbulent viscous reacting flows and their interaction with the engine structural components.

TECHNICAL APPROACH

The technical approach is to develop a numerical simulation of the unsteady operation of a gas turbine engine. This includes the steady and unsteady flow through the airfoil rows, the combustor flow field, and the interaction between the airfoil row and combustor flow fields, i.e., a time-varying coupled analysis of the combustor and turbomachinery components.

Turbomachinery Aerodynamics & Blade Row Fluid-Structure Interactions

Classical Approach

The primary mechanism of blade failure is fatigue caused by vibrations at levels exceeding material endurance limits. The classical approach to predict the amplitude responses and stresses at the resonant speeds is indicated in Figure 5.

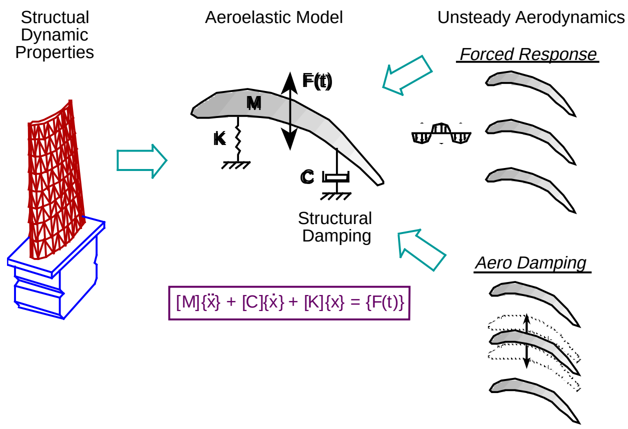

Figure 5. Classical approach to fluid-structure interactions

The structural properties of both rotating or stationary airfoil rows are predicted first with finite element models. To determine the unsteady aerodynamic loading on the airfoil rows, a definition of the unsteady aerodynamic forcing function in terms of its harmonics is required. The forcing function is then Fourier decomposed into harmonics. The unsteady aerodynamic response of the airfoil row to each forcing function harmonic is then assumed to be comprised of two components: the disturbance being swept past the nonresponding airfoils, termed the gust unsteady aerodynamics; the airfoil vibratory response to this disturbance, the motion-induced unsteady aerodynamics or the aerodynamic damping. Finally, an aeroelastic or structural dynamics model is utilized to predict the vibratory response of the airfoil row.

In this classical approach, the fluid and structure are modeled separately, i.e., they are not coupled. They are then coupled by specifying the kinematic boundary conditions at the fluid-structure boundary. Thus, the unsteady aerodynamic forces acting on the blading are predicted with the motion of the structure as a boundary condition. Such unsteady flow models assume that (1) the unsteadiness is a small perturbation from the steady flow and (2) the airfoil motion is specified.

Unfortunately, this uncoupling of a truly coupled problem introduces error into the simulations. Namely, the blade row unsteady aerodynamic loading is dependent on the specified motion of the blade. Clearly, this is not a fully coupled fluid-structure interaction problem. Thus, instead of utilizing separate fluid and structural models, a coupled interacting fluid-structures analysis is needed. The increasing computational capabilities now available make a coupled interacting fluid-structures analysis feasible.

In this regard, the finite element code ALE3D (Arbitrary Lagrange/Eulerian 3D Code System) developed at Lawrence Livermore National Lab is a finite element code which has been applied to analyze various metal forming processes. Its formulation is general enough to model both fluids and structures in a Lagrangian, an Eulerian, or an intermediate perspective.

Research Plan

A next generation coupled fluid-structure interaction model for turbomachine blade rows will be developed. This will be accomplished by extending ALE3D to address the unsteady aerodynamics of turbomachine blade rows. A schematic of the blade row flow field simulation utilizing ALE3D is presented in Figure 6. Inputs include the blade row geometry, the inlet flow field and the mass flow, specified by means of the blade row exit pressure. The output includes the blade row inlet flow field, iterated with the input to meet the mass flow requirement, and the blade row flow field, including the exit flow field. Note that this exit flow field is the combustor inlet flow field.

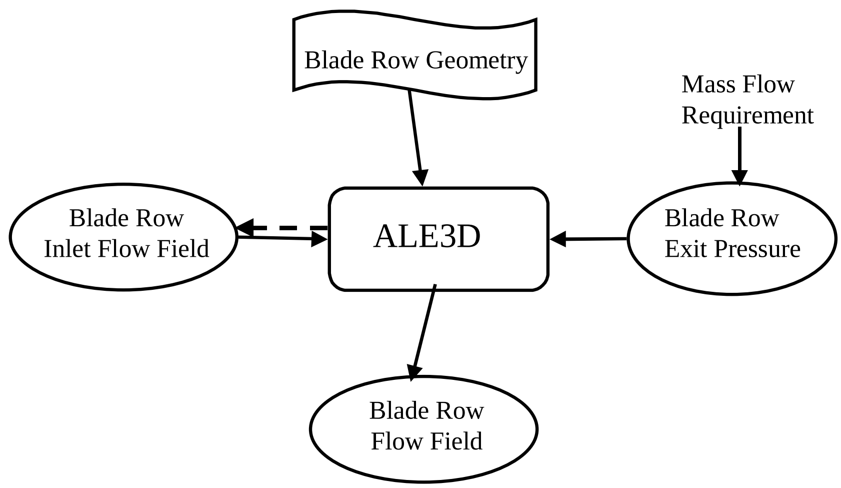

Figure 6. Blade row flow field - ALE3D

ALE3D will first be extended to predict the steady and unsteady aerodynamics of a turbomachine blade row. This requires the use of inflow and outflow boundary conditions and appropriate turbomachine blade and flow mesh algorithms. The resulting finite element Euler analysis of cascade steady and unsteady aerodynamics will then be verified for both subsonic and transonic flow conditions. This will be accomplished by comparing ALE3D predictions with appropriate predictions from other models for simplified geometries and also correlating with experimental data

The issues associated with transient gas turbine engine performance will then be addressed. This will initially accomplished by applying a time-varying exit flow boundary condition to simulate rotating stall and surge conditions, with the resulting blade row unsteady loading predicted. In this case the geometry will be based on that of an actual engine.

Combustor Flow Field

The gas turbine engine combustor increases the enthalpy of the working fluid by oxidization of fuel and the subsequent dilution of the resulting products with additional air until temperatures acceptable to the turbine are achieved. From an operability viewpoint, the combustor must provide a flow environment that is conducive to both ignition and the stability of the flame over a wide variety of engine operating conditions. To meet steady performance requirements, acceptable exit temperature profiles, low-pressure losses and minimal pollutant emissions are necessary.

CFD gas turbine combustor modeling has generally been limited to isolated parts of the combustion system. Most models include only the reacting flow inside the combustor liner with assumed profiles and flow spits at the various liner inlets. Carefully executed models of this type can provide valuable insight into mixing performance, pattern factor, emissions and combustion efficiency.

A CFD calculation for the unsteady flow through a complete annulus combustor - from the compressor diffuser exit to the turbine inlet - is needed. The model should include an airblast fuel nozzle, dome and liner walls with dilution holes and cooling louvers.

Research Plan

The combustor will be modeled by the KIVA Code. This analysis, originally developed to predict the combustion processes in diesel engines, will be implemented for application to gas turbine engine combustors. A schematic of the combustor simulation utilizing KIVA is presented in Figure 7. Inputs include the combustor geometry, the inlet flow field and the mass flow, specified by means of the combustor exit pressure. The output includes the combustor flow field and performance, iterated with the input to meet the mass flow requirement, and the combustor flow field, including the inlet flow field. Note that this inlet flow field is the blade row exit flow field.

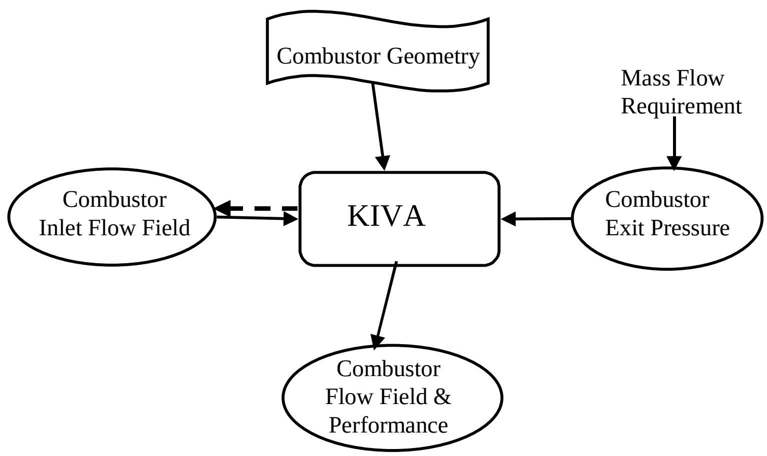

Figure 7. Combustor analysis - KIVA

The steady performance of a combustor will be predicted first. The issues associated with transient gas turbine engine performance will then be addressed. This will initially be accomplished by applying time-varying inlet flow conditions to the combustor, with the resulting time-varying combustor performance predicted

Simulation of Gas Turbine Engine Transient Performance

The advanced simulation of transient gas turbine engine performance will also be addressed. This simulation will capture the important compressor-combustor interactions that occur during engine transients, including rotating stall and surge, and their affect on blade row durability. This will be accomplished by coupling the unsteady flow field analysis ALE3D and the combustor model KIVA. This coupled simulation is depicted in Figure 8. The mass flow requirement is met through the combustor exit pressure condition. The blade row and combustor simulations are coupled through the ALE3D predicted blade row exit flow field that is the combustor KIVA input and the KIVA predicted combustor inlet flow field which is the blade row exit pressure.

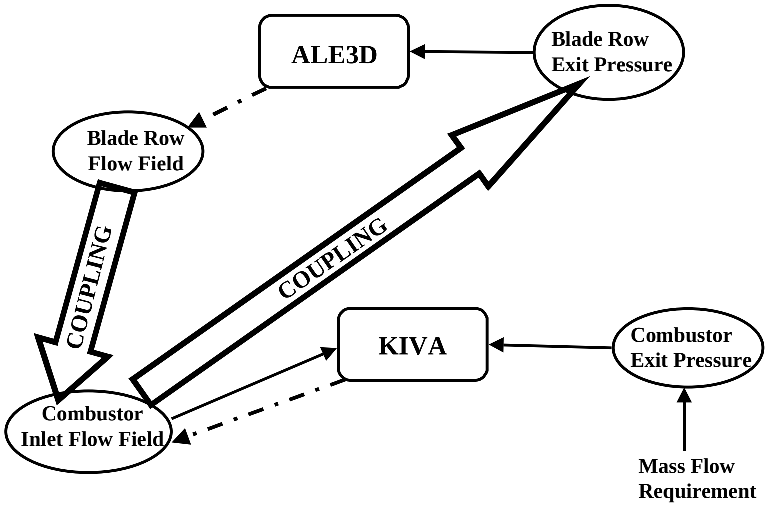

Figure 8. Coupled gas turbine engine transient simulation schematic

ACKNOWLEGEMENT

This work is supported by the U.S. Department of Energy ASCI program, contract LG-6982. The author's addresses are: Fleeter, Dept. of Mechanical Engineering, W Lafayette, IN campus; Houstis and Rice, Dept. of Computer Sciences, W Lafayette, IN campus, Zhou, Dept. of Mechanical Engineering, Calumet, IN campus.