Annex A – HIGHER ORDER MODELS

INTRODUCTORY COMMENTS

This Annex provides the reader with an insight into a whole category of simulation techniques that have been developed and continue to be developed for phenomena that have been observed in gas turbine engines but can't be simulated within the 0-D cycle model concept. The 1-D dynamic models used for compression system and engine operability (such as surge and rotating stall) are especially highlighted and several examples have been given to indicate how these models have been used.

A.1 DETAILED 1-DIMENSIONAL (1-D) MODELS

A 1-D model typically tries to extend a 0-D cycle model representation by physically modeling an additional dimension. The most commonly used 1-D models are for dynamic simulations where 0-D models require the addition of a length dimension to adequately model high frequency behavior. The next most common use is where components or local areas of the engine are modeled at a higher level of fidelity in either the radial, circumferential or axial direction to address specific concerns. Often these models remain at 0-D fidelity in other parts of the engine. 0-D and 1-D models may be used predict 2-D and 3-D variations in engine conditions and may use a physical model to do it. But the physical models stay at the lesser level of detail. An example is the use of a more detailed compressor model, which predicts the variation in driving pressures at the hub and tip, and then uses these to model internal leakage and cooling circuits.

A.1.1 Steady State 1-D Models

Steady state 1-D models are typically used to provide detailed information inside one or more components such as:

- Blade row, cooling passage or cavity level information. In some cases this may not be a true 1-D model but merely a 0-D model taken to a greater than normal level of detail.
- Tracing radial or circumferential information through the engine. This could be inlet pressure or temperature distortion, water or fuel cloud, or a non-uniform condition created inside the engine. This might be in the burner due to a fuel nozzle variation, or in a compressor due to a locally off-angle sector of stators.

When providing more detail inside a component, more detailed boundary conditions may not be required. For models addressing radial or circumferential behavior through the engine, creation and transfer of the additional boundary conditions is the key part of transient 1-D models.

A.1.2 Transient 1-D Models

Often these models are identical to a steady state 1-D model and only include the transient extensions found in 0-D models. This includes models of low frequency phenomena such as rotor accelerations, heat transfer and active or passive geometry changes. Active geometry changes modeling can include dynamics associated with variable stators, variable bleed-valves, or modulated cooling flows such as anti-ice air or variable exhaust nozzles. Passive geometry changes can include turbine or exhaust nozzle areas, which change with temperature or tip clearance. Heat transfer effects must typically cover a range of time constants (from a few seconds to a few minutes) to accurately model engine behavior. These models may include volume dynamics to address leakage or cavity flows or to meet other accuracy needs. However, these models are generally not intended to address acoustic level phenomena such as stall, surge or combustion instability.

A.1.3 1-D Dynamic Engine Simulations

The purpose of this section is to describe recent advances in modeling gas turbine engine dynamic behavior. A dynamic turbine-engine combustor model and simulation, VPICOMB, which was developed at Virginia Polytechnic Institute and State University, [A.1] will be discussed. The integration of the VPICOMB combustion model equations with the DYNamic Turbine Engine Compressor Code (DYNTECC), conducted at the Arnold Engineering Development Center (AEDC), [A.2 and A.3] will then be discussed. Finally, a full gas turbine engine model and simulation, the Aerodynamic Turbine Engine Code (ATEC), also developed at AEDC, [A.4 and A.5] will be described. ATEC currently has the ability to simulate a turbojet engine with the compressor system operating post-stall, and a turbofan engine operating up to the point of compressor stall.

The governing equations are derived by the application of mass, momentum, and energy conservation to the elemental control volume where:

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}}{\partial x} = \mathbf{G} \qquad \textbf{Eq. A-1}$$

where:

$$\mathbf{U} = \begin{bmatrix} A\rho \\ \rho Au \\ AE \end{bmatrix} \quad \mathbf{F} = \begin{bmatrix} \rho Au \\ \rho Au^2 + AP \\ u(AE + AP) \end{bmatrix} \quad \mathbf{G} = \begin{bmatrix} -W_{B_x} \\ FX_x \\ Q_x + SW_x - H_{Bx} \end{bmatrix} \qquad \textbf{Eq. A-2}$$

Additionally, a perfect gas is assumed and the equation of state,

$$P = \rho RT \qquad \textbf{Eq. A-3}$$

is used. The assumption of constant specific heats (Cp) and ratio of specific heats (γ) in the property calculations, while computationally efficient, does result in predicted temperature levels in the combustor that are too high during rich combustion. This effect will be shown in the following sections. The distributed turbomachinery source terms $-w_{B_x}, F_x, Q_x + SW_x - H_{Bx_x}$ are supplied to the overall model by the user. Models for the source terms will be discussed in each of the respective sections.

The time dependent flow-field within the system of interest is obtained by solving the time dependent system of equations using one of two numerical approaches. The VPICOMB model [A.1] uses an explicit Roe's flux-differencing scheme adapted to the Euler equations with source terms to evaluate the face fluxes, and then uses a second-order four step Runge-Kutta algorithm to solve for the time dependent equations. DYNTECC [A.2 and A.3] and ATEC [A.4 and A.5] use a flux-difference-splitting scheme based upon characteristic theory to solve for the face fluxes. A first order Euler method is used to solve for the time dependent equations.

A.1.3.1 Combustor Component – VPICOMB

The VPICOMB model and simulation provides a one-dimensional tool running on the personal computer for analyzing dynamic gas turbine engine combustor operation. With VPICOMB, the user can analyze the influence of varying inlet conditions, fuel pulses, exit flow restrictions, and many other possible dynamic events. The user provides the simulation with appropriate initial and boundary conditions, plus any time dependent variations in the boundary conditions, and then exercises the program to determine the time dependent flow-field.

For the inlet boundary conditions, the user specifies inlet total pressure, total temperature, and flow rate. The user can also specify the friction factor along the wall of the combustor. No pressure loss due to combustion is assumed. The exit boundary condition assumes a constant value for the mass flow parameter:

$$\frac{W\sqrt{T_T}}{P_T A} = \text{Const} \qquad \textbf{Eq. A-4}$$

The value of the mass flow function is obtained during the initial condition calculations. It is assumed that the fuel mass flow addition occurs in the first control volume. The heat release due to combustion occurs in the control volumes specified by the user as the zone of heat release. The heat release is equally distributed across the zone of heat release control volumes. The amount of energy released in the zone of heat release control volumes is a function of fuel flow rate, the combustion efficiency, the lower heating value of the fuel, and the combustor flammability limits.

The combustor flammability limits are determined by using steady state engineering correlations developed by Herbert, 1957. In order for stable combustion to occur, the primary zone equivalence ratio (ϕ_{PZ}) must fall within a rich and lean limit:

$$\phi_L \leq \phi_{PZ} \leq \phi_R \qquad \textbf{Eq. A-5}$$

Based on experimental data, Herbert defined a Combined Air Loading Factor to calibrate the light off and blow off data. A polynomial curve fit of Herbert's flammability data for a generic can type combustor is used in the VPICOMB model. Combustion efficiency is determined by using steady state engineering correlations developed by Lefebvre,1985 [A.6]. Lefebvre assumed that the efficiency of fuel evaporation and the reaction efficiency limit the overall combustion efficiency. Further modification to the Lefebvre work was done by Derr and Mellor, 1990 [A.7].

Because of the dynamic operation of the combustor, it is possible for heat release to occur for a short period of time even though the combustor equivalence ratio may lie outside the steady state flammability limits. Likewise, the heat release process may not resume immediately after the combustor equivalence ratio re-enters the flammability bounds. To account for these effects, a first order lag on the heat release rate has been incorporated in the model:

$$\tau \frac{d\mathring{Q}}{dt} + \mathring{Q} = \mathring{Q}_{ss}(t) \qquad \textbf{Eq. A-6}$$

A.1.3.2 Compressor Component – DYNTECC

DYNTECC [A.2 and A.3] is a one-dimensional, stage-by-stage, compression system mathematical model that is able to analyze any generic compression system. DYNTECC uses a finite difference numerical technique to simultaneously solve the mass, momentum, and energy equations with turbomachinery source terms (mass bleed, blade forces, heat transfer, and shaft work). The source terms are determined from a complete set of stage pressure and temperature characteristics provided by the user.

Illustrated in **Figure A.1** is a representative, single-spool, multi-stage compressor and ducting system. An overall control volume models the compressor and ducting system. Acting on the fluid control volume is an axial-force distribution, FX, attributable to the effects of the compressor blading and the walls of the system. Appropriate inlet and outlet boundary conditions are applied at the inflow and outflow boundary

locations. Energy supplied to the control volume includes the rate of heat added to the fluid, Q, and shaft work done on the fluid, SW. Mass transfer rates across boundaries other than the inlet or exit, such as the case of inter-stage bleeds, are represented by the distribution, W_B.

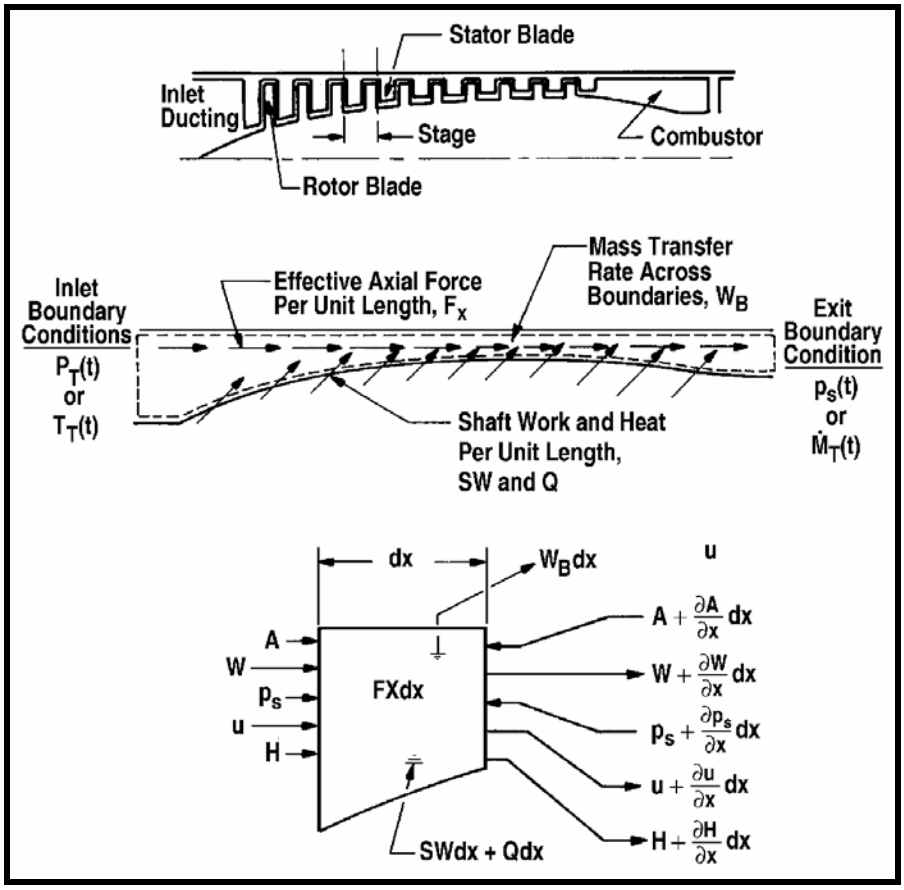


Figure A.1: DYNTECC Control Volume Technique.

The overall control volume is subdivided into a set of elemental control volumes. Typically, the compressor section is subdivided by stages either as rotor-stator or vice versa depending on the way experimental stage characteristics may have been obtained. All other duct control volumes are divided to ensure an appropriate frequency response. The governing equations are derived from the application of mass, momentum, and energy conservation principles to each elemental control volume.

To provide stage force, *FX*, and shaft work, *SW,* inputs to the momentum and energy equations, a set of quasi-steady stage characteristics must be available for closure. The stage characteristics provide the pressure and temperature rise across each stage as a function of steady airflow. Using pressure rise, temperature rise, and airflow, a calculation can be made for stage steady-state forces and shaft work. A typical set of stage characteristics is presented in **Figure A.2**.

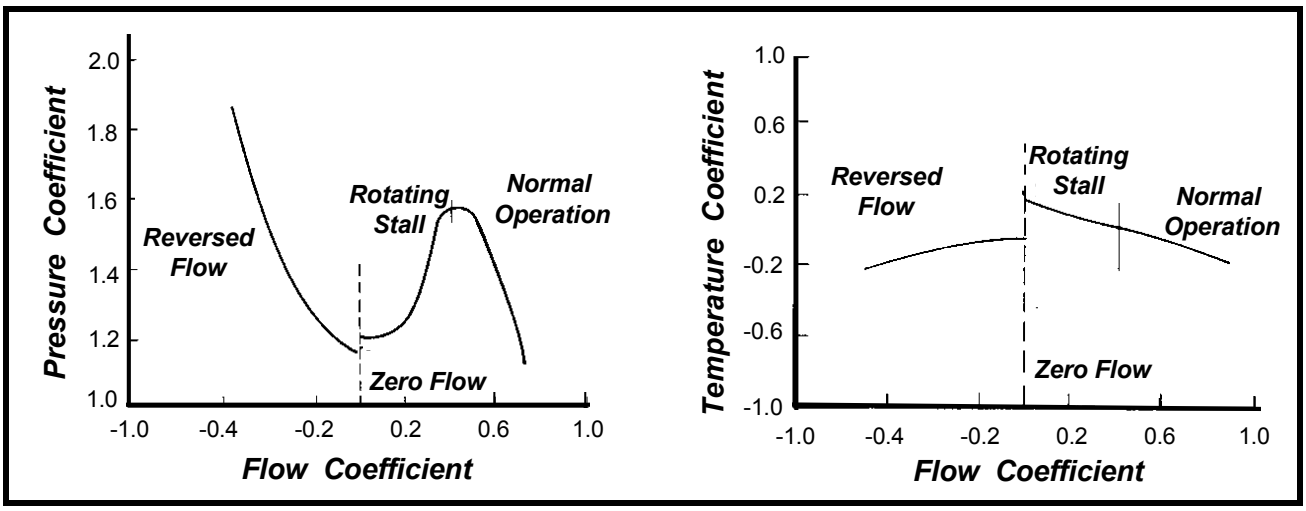


Figure A.2: Typical Set of Stage Characteristics.

The above discussion centers on the steady characteristic. During transition to surge and development of rotating stall, the steady stage forces derived from the steady characteristics are modified for dynamic behavior via a first-order lag equation of the form:

$$\tau \frac{d(FX)}{dt} + FX = FX_{ss} \qquad \textbf{Eq. A-7}$$

The time constant, τ, is used to calibrate the model to provide the correct post-stall behavior. The inflow boundary during normal forward flow is the specification of total pressure and temperature. The exit boundary condition is the specification of exit Mach number or static pressure. During reverse flow the inlet is converted to an exit boundary with the specification of the ambient static pressure. Therefore, both the inlet and the exit boundary function as exit boundaries during a surge cycle.

An explicit split-flux finite-difference algorithm is used to numerically solve the area weighted quasi-one-dimensional Euler equations. The quasi-one-dimensional Euler equations with source terms (**Eq. A-1** and **Eq. A-2**) are written in conservation Cartesian form and applied to a fixed grid. A finite difference representation of **Eq. A-1** can be applied over an interval between grid points j and j+1 with the fluxes evaluated at the nodes and the sources evaluated at the center of the volume given by

$$\left(\frac{\Delta U}{\Delta t}\right)_j = I^+_{j-\frac{1}{2}}\left(\frac{\Delta U}{\Delta t}\right)_{j-\frac{1}{2}} + I^-_{j+\frac{1}{2}}\left(\frac{\Delta U}{\Delta t}\right)_{j+\frac{1}{2}} \qquad \textbf{Eq. A- 8}$$

where

$$\left(\frac{\Delta U}{\Delta t}\right)_{j-\frac{1}{2}} = \left[G_{j-\frac{1}{2}} - \frac{(F_j - F_{j-1})}{(x_j - x_{j-1})}\right]; \qquad \left(\frac{\Delta U}{\Delta t}\right)_{j+\frac{1}{2}} = \left[G_{j+\frac{1}{2}} - \frac{(F_{j+1} - F_j)}{(x_{j+1} - x_j)}\right] \qquad \textbf{Eq. A- 9}$$

Characteristic theory is used to develop weighting terms (I^+, I^-) for splitting the time derivatives to the adjacent nodes as illustrated in **Figure A.3**.

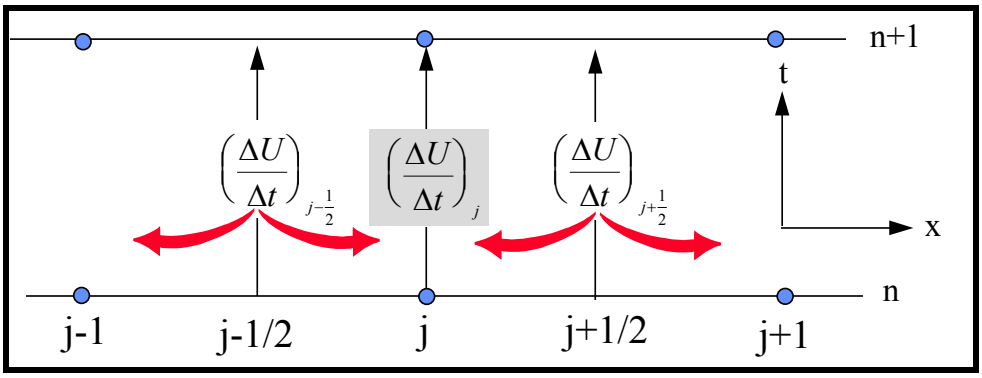


Figure A.3: Schematic of DYNTECC Explicit Split Flux-Differencing Scheme.

The time derivatives at the nodes can be obtained by summing the left characteristic weighted time derivative from an upstream interval and the right characteristic weighted time derivative from the downstream interval. A solution is now obtainable at the n+1 time step by a forward Euler time integration procedure.

When circumferential inlet distortion effects are important, DYNTECC can be operated as a parallel compressor model with or without circumferential and radial cross-flow approximations. This is illustrated in **Figure A.4**. Modified parallel compressor theory (Shahrokhi, 1995), **[A.8]** has been applied to permit the simulation of dynamic inlet distortion. The overall compression system control volume is sub-divided into a series of circumferential and parallel tubes. Each segment or tube then acts in parallel with each other segment, exiting to the same exit boundary condition. Different magnitudes of inlet total pressure and temperature can then be imposed upon each segment of the parallel compressor. In the purest sense, each segment is independent of all other segments, except through the exit boundary condition. For complex distortion patterns the circumferential and radial cross-flow terms are approximated. System instability occurs when any one segment becomes unstable as a result of the inlet and exit conditions imposed upon it.

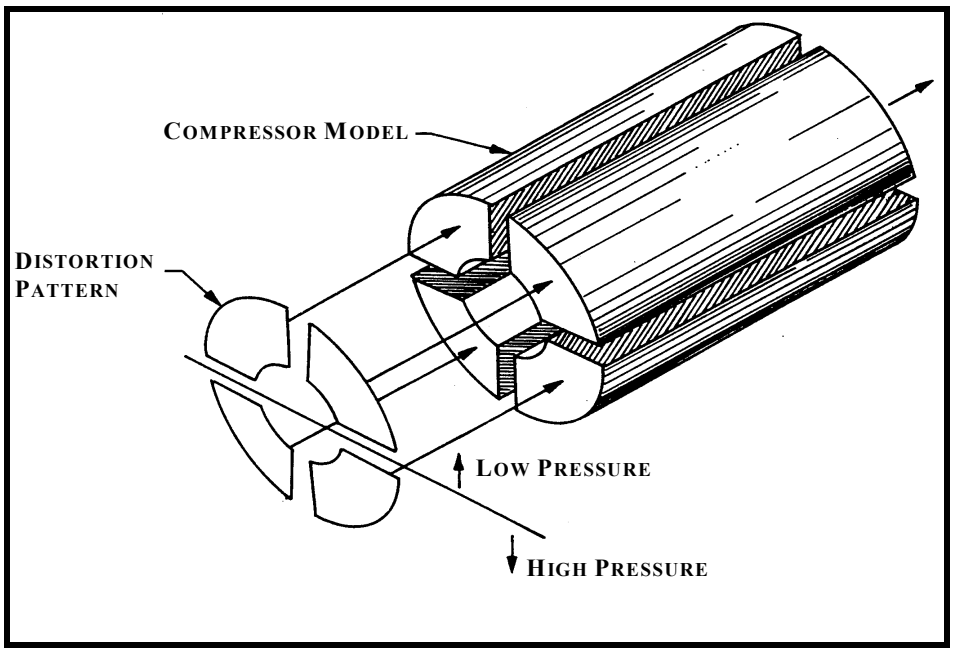


Figure A.4: Compressor with Circumferential Segments and Applied Inlet Distortion.

Parallel compressor theory is generally valid if the segment arc is greater than 60 degrees, also known as the critical angle. Secondary flow mechanisms become more significant for segments with arcs less than the critical angle. The parallel compressor theory's predictive capabilities deteriorate when segments of less than the critical angle are used.

Both pressure and temperature characteristics are required. During pre-stall operation, the steady state compressor characteristics are used as given. During post-stall operation, the change in compressor operating conditions is lagged using the same first order equation used in lagging the VPICOMB combustion heat release rate. The most recent version of DYNTECC has been upgraded to include the VPI developed combustor model discussed above. This new feature will permit the user to study dynamic compressor and combustor interactions.

A.1.3.3 Full Engine Simulation – ATEC

With the inclusion of the VPICOMB combustor model equations into the DYNTECC program, development of a gas turbine engine model and simulation required was completed with the addition of a turbine model. Because of the modularity of the DYNTECC coding, integrating the turbine model into the existing code required specifying the performance characteristics in a fashion similar to the compressor performance (shaft work and blade forces) and as shown in **Figure A.5** and compared to a component level modeling scheme.

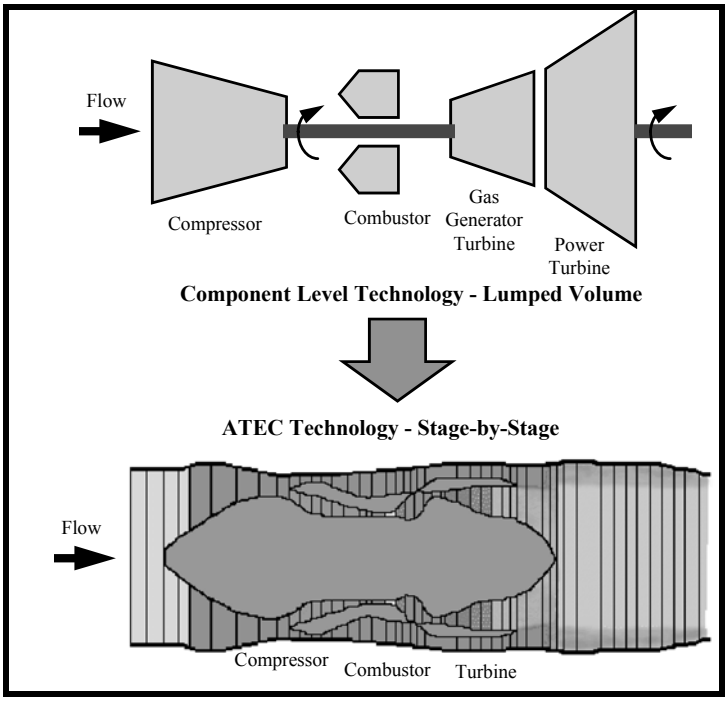


Figure A.5: ATEC Code Compared to Component Level Modeling Technique.

ATEC (Garrard, 1996), **[A.4** and **A.5]** is currently configured to support two gas generator turbines coupled to compressor systems through up to two shafts, and one power turbine. To determine the amount of work extracted across a given turbine during the initial condition calculations, the gas generator turbine is assumed to exactly provide the power required by the compressor system. The user initially specifies work output from the power turbine. Once the time integration starts, energy extraction is given by the pressure ratio across the turbine and the turbine inlet flow function.

Operational demonstration of the dynamic engine model has been accomplished by using the T-55 turboshaft and the J-85 turbojet engines. Characteristics for each of the turbines are overall, not stage-by-stage, due to lack of inter-stage data. The shaft work and blade forces in each turbine were equally distributed across multiple control volumes, however, to keep the overall length of any given control volume on the same order as the rest of the grid. This helps maximize model dynamic fidelity and, with the explicit flow solver routine, numerical stability.

Steady state results of the engine system are compared to the engine manufacturer's steady state engine model. The comparison results are shown in **Figure A.6**. Close agreement between the two models was obtained. It is judged that the majority of the differences can be attributed to the fact that the compressor characteristics for ATEC were based on compressor rig data, which was not the same as used in the steady state model. Even with this difference, the maximum error was less than seven percent.

	Total Pressure (P/P_{ref})			Total Temperature (T/T_{ref})		
Location	**Mfg**	**ATEC**	**%Delta**	**Mfg**	**ATEC**	**%Delta**
Inlet	0.29	0.29	0.00	0.26	0.26	0.00
Compressor Exit	2.38	2.36	0.73	0.51	0.51	0.72
Burner Exit	2.29	2.25	1.76	1.16	1.15	0.47
Gas Generator Turbine Exit	0.81	0.87	-6.27	0.93	0.96	-3.63
Power Generator Turbine Exit	0.30	0.30	-0.75	0.75	0.79	-5.22

Figure A.6: Comparison of ATEC and Engine Model Results.

To demonstrate dynamic operation of ATEC, the same test case as was used to demonstrate the integration of the VPICOMB model equations into DYNTECC has been exercised.

Variation of the relative total pressure in the engine is shown in **Figure A.6**. During the initial steady state operation, the total pressure increases through the compressor system. In the combustor, a small total pressure loss occurs. Work extraction in the turbines reduces the pressure back to near atmospheric before the flow exits the engine. As with the DYNTECC test case, the fuel flow pulse forces the compressor into surge. Rather than being driven by a constant Mach number exit boundary condition, however, the pressure increase is tied to the turbine choking as explicitly defined by the turbine steady state operating characteristics. Steady state operation is not re-established until the fuel flow rate is decreased back to the original flow rate. The frequency of the surge cycles is reduced due to the increased volume of the calculation domain.

Relative total temperature in the engine as a function of time is also shown in **Figure A.6**. As the compressor enters the surge cycle, the reduction of air mass flow rate causes the combustor temperature to increase dramatically, until the equivalence ratio rises to the rich flammability limit. After the surge cycle

is completed, the combustion process is re-established until the next cycle forces the equivalence ratio to rise above the flammability limit.

Relative mass flow rate in the engine as a function of time is shown in **Figure A.7**. In addition to the bleed extraction occurring in the axial compressor and the fuel addition occurring in the combustor, turbine-cooling bleed is injected into the flow in the last gas-generator turbine control-volume. As with the DYNTECC test case, during each of the surge cycles, the mass flow rate in the front section of the engine reverses and becomes negative. In the back section of the engine, the flow rate is greatly reduced, but it does not reverse.

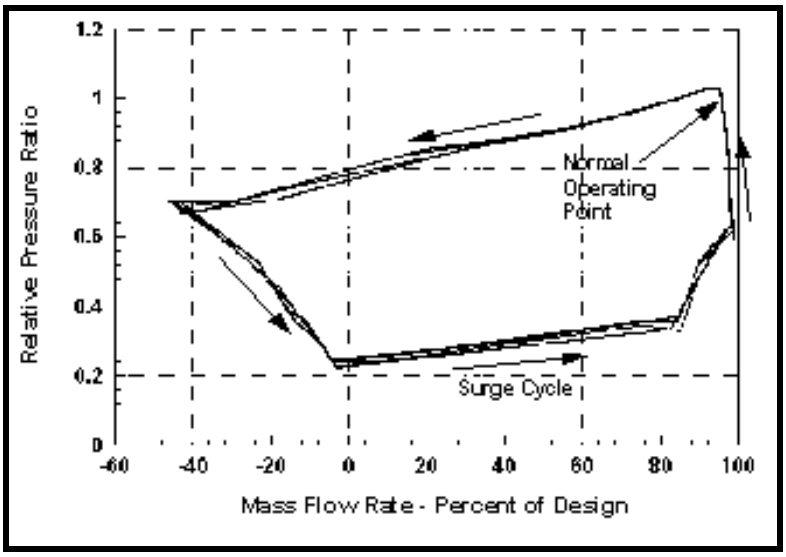


Figure A.7: System Response with Small Lag.

To demonstrate the capability of the model and simulation to simulate rotating stall in the compressor system, the above test case was repeated with a modification to the time-lag constant used with the compressor characteristics in the rotating stall regime. By increasing the time constant of each stage of the compressor system, the operational characteristics change. This results in the compressor system being unable to return to the normal operating mode. The effect is shown by comparing the compressor pressure ratio curves as a function of mass flow rate through the system for both cases. This comparison is shown in **Figure A.8**. With a relatively small time constant, the dynamics of the system allow the compressor to return back to the normal pre-stall operating speed line before re-entering the surge cycle, as is shown in **Figure A.7**. As is shown in the various figures, once the perturbation in fuel flow rate is removed, the system returns to normal, steady state operation. An increase in the compression system time constants forces the system into rotating stall, as is shown in **Figure A.8**. Even when the fuel flow rate is reduced back to the original level, the system is unable to recover back to the original steady state operating point.

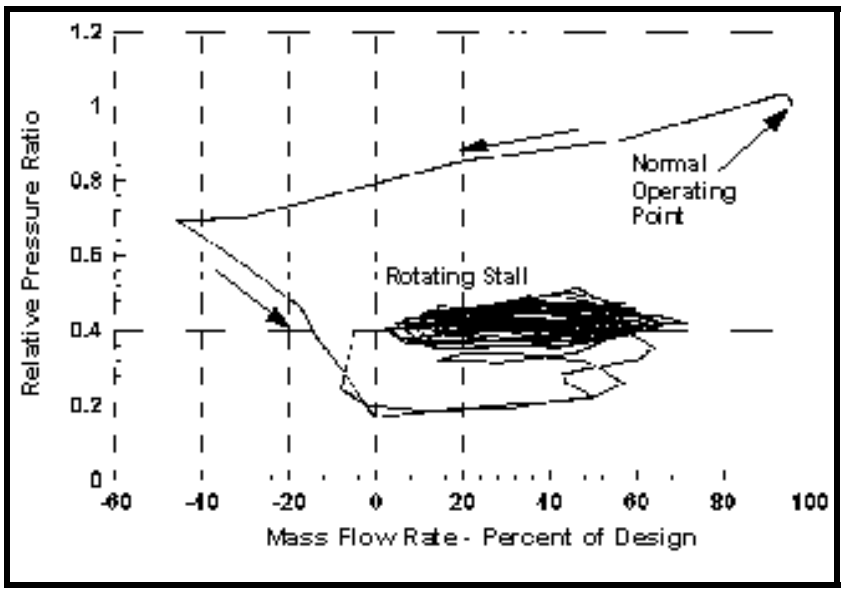


Figure A.8: System Response with Large Lag.

A.1.4 Benefits of 1-D vs. 0-D Models

1-D models provide additional boundary condition details not covered by 0-D models. The reason for using a more detailed model in place of a purely empirical correlation based on 0-D conditions is to obtain greater accuracy or better fidelity in response to changing conditions. Correlations to extend 0-D models are often difficult to create and require significant effort to recreate if engine operating conditions or component representations change.

Detailed 1-D models are generally well adapted to desktop PC and UNIX workstations. They do not represent a significant computational barrier unless near real-time transient performance is required or they are to be part of a larger simulation or a study requiring a large number of runs.

A.2 HIGH FIDELITY 2-D/3-D MODELS

2-D and 3-D models not only predict but also attempt to model the physics at this higher level of detail.

0-D or 1-D models may be used predict 2-D and 3-D variations in engine conditions, and may use a physical model to do it, but the physical models stay at the lesser level of detail. Engine system level models at this level of detail are just becoming practical from both computation and component modeling needs.

Detailed models provide potential for closer modeling of the physics, reducing the need for empiricism and to provide insight into the component operation and variation. The primary near term benefit of using 2-D and 3-D models of an engine system is to provide insight into the component operation details that 0-D and 1-D models cannot.

A.2.1 Zooming of 2-D Models

2-D Axisymmetric models have been the workhorses for compressor and turbine design for many years. 3-D tools are gradually replacing them. The primary benefit of using 2-D system models is to facilitate the design and detail analysis of a particular component across the range of possible boundary conditions. In the past, running lower fidelity models did this. More detailed boundary conditions were created through

an empirical process, and then the 2-D code was run in isolation. Computing capacity made this process the only reasonable option.

It is now possible to run a 2-D component simulation, in conjunction with a simplified model of the rest of the engine, in a few minutes and to even run the entire engine in less than an hour. The choice now falls between:

- Modeling a component or the entire engine at the 2-D level, which must be justified in terms of the extra model development complexity and effort required to add value to the results; and
- Obtaining estimates that can be made based on a simpler model.

Currently, it is only in the design and detailed insight into the turbo-machinery where the added information justifies the more detailed modeling. The engine level performance impact of this greater detail is generally lost in the overall empiricism and modeling uncertainty. For example, 2-D modeling of clearance effects can be of interest in the analysis and design of the compressor. However, the predictive accuracy and stability of currently available models does not justify their use in a whole engine model. **Figure A.9** indicates typical characteristics for different types of model.

Model Type	Accuracy	Stability	Computation Speed
Component Design	High	Low	Low
Component Analysis	Medium	Medium	Medium
Engine Analysis	Low	High	High

Figure A.9: Model Attributes for Design and Analysis.

A.2.2 3-D Models (Euler and RANS)

Full physics 3-D CFD models for single blade rows and components are beginning to see use outside of research areas. For system models, virtually all require some level of empiricism or simplification such as use of source terms or body forces for the turbo-machinery or an average passage or mixing plane assumption.

Two types of 3-D codes may be encountered: Euler and Reynolds Averaged Navier-Stokes (RANS). Examples of Euler codes include the dynamic Turbine Engine Analysis Compressor code, TEACC, **[A.9]** and the steady state NASA ENG10 **[A.10]** code. These codes generally use some modeling simplifications (source terms, streamline curvature methods) to allow solution on a single workstation in minutes to hours. They can generally run multiple points or even transients. As computing power grows, these limitations may be reduced. However, the need to accept simplifying assumptions to allow for the desired level of full engine analysis will remain for some time.

An example of RANS codes are: The NASA Average Passage Code (APNASA) **[A.11]** is an example of a component code which has been extended to cover all the blade rows in an engine. Simplified component models or boundary conditions inputs are used to address portions of the engine that can not be addressed using the component code. In some instances two multi-dimension components codes are joined to address the interaction between different component types. The NPARC and ADPAC code have been combined to study Fan and Inlet interaction **[A.12]**. Models based on combining or extending detailed component codes require the highest level of computer resources. To run a single point, these codes may

require in dozens or even hundreds of workstations. Codes have been developed for the purpose of multi-dimensional simulation throughout the engine.

Three-dimensional codes require a level of geometry detail not generally required for 0-D system performance models. The detailed component models often require gridding and application to different disciplines may put additional compatibility constraints on the solution. Often the bookkeeping of the geometry and gridding exceeds the complexity of other parts of the simulation. This process is often facilitated by use of a CAD package to manage geometry and geometry centered information. Linking this information to a CAD system designed to address these issues greatly simplifies the already complicated job of creating and managing the simulation. Some more details on these types of codes can be found in ***Annex B***.

A.3 HIGHER ORDER MODELING APPLICATIONS

Included in this Annex are synoptics or examples of applications using higher order (1-D, 2-D and 3-D) component or engine simulations. The format established in ***Chapter 2*** has been retained to provide the reader with a sense of what the code can do and an applicable reference for further enlightenment.

A.3.1 1-D Dynamic Model Applications

An important component of the gas turbine engine is the compression system. In today's military turbine engines, the compression system consists of one or more axial compressors. These axial compressors must operate in a stable manner even with severe inlet pressure or temperature distortion. Many experimental and analytical investigations have been conducted in the past three or four decades to separately quantify the effects of pressure or temperature distortion on the compression system. While little experimental work has been performed on combined time-variant total pressure and temperature distortions, some investigations have been carried out to examine the effects of combined steady-state pressure and temperature distortion on compression system stability. With the advent of highly agile maneuvering aircraft with weapons release near the engine inlets, there exists a requirement to quantify the combined effects of severe pressure and temperature distortion, both transiently and in the steady state.

A.3.1.1 Rotating Stall and Surge Investigation Using 1-D Compressor Model

In most aircraft gas turbine engines, the compression system consists of one or more aerodynamically coupled axial-flow compressors. It is the function of the compression system to increase the static pressure and density of the working fluid. Without stable aerodynamic operation, the compression system cannot deliver the desired increase in static pressure and density. During operation of axial-flow multi-stage compression systems in gas turbine engines, undesired system phenomena known as surge and rotating stall have been observed. Several modeling techniques have been developed over the last decade to investigate surge and rotating stall. One technique involves a stage-by-stage one-dimensional approach described below.

Modeling Techniques Used

DYNTECC is a one-dimensional, stage-by-stage, compression system mathematical model, **[A.2** and **A.3]**, which is able to analyze any generic compression system (see ***Section A.1.3.2***). DYNTECC uses a finite difference numerical technique to simultaneously solve the mass, momentum, and energy equations with turbomachinery source terms (mass bleed, blade forces, heat transfer, and shaft work). The source terms are determined from a complete set of stage pressure and temperature characteristics provided by the user.

Potential Benefits

Mathematical compressor models have become a major tool for understanding compression system behavior during dynamic events, such as inlet distortions. A validated compression system model can be used to extend the range of the experimental test results to the untested regime. Using a validated stage-by-stage compression system model, a parametric investigation can be conducted to determine the qualitative effects of various recovery actions or design changes on system operability.

Cited Examples

- Hale, A.A. and Davis, Jr., M.W., "Dynamic Turbine Engine Compressor Code, DYNTECC – Theory and Capabilities", AIAA Paper # 92-3190, June 1992. **[A.2]**
- Davis, Jr., M.W. et al., "Euler Modeling Techniques for the Investigation of Unsteady Dynamic Compression System Behavior", Loss Mechanisms and Unsteady Flows in Turbomachines, AGARD-CP-571, January 1996. **[A.3]**

DYNTECC was configured to aid in the analysis of a ten-stage compressor rig test conducted at the Compressor Research Facility to investigate the boundary between surge and rotating stall for high-pressure ratio compressors. The model was executed at the experimentally determined stall and surge boundary where the recovery hysteresis was most severe. DYNTECC was calibrated at the stall and surge boundary by adjusting calibration constants to produce rotating stall. The constants were held fixed for subsequent analysis that investigated possible hardware modifications to reduce the sensitivity of the compressor to rotating stall.

Calibration of DYNTECC with experimental data matched the overall performance and individual stage performance of the 10-stage compressor rig. As illustrated in **Figure A.10**, the compressor initially experiences a partial surge cycle, then transitions to rotating stall. DYNTECC individual stage performance is also compared to that obtained experimentally. For both overall and individual stage behaviors, DYNTECC reproduced the experimental results fairly accurately during the dynamic event.

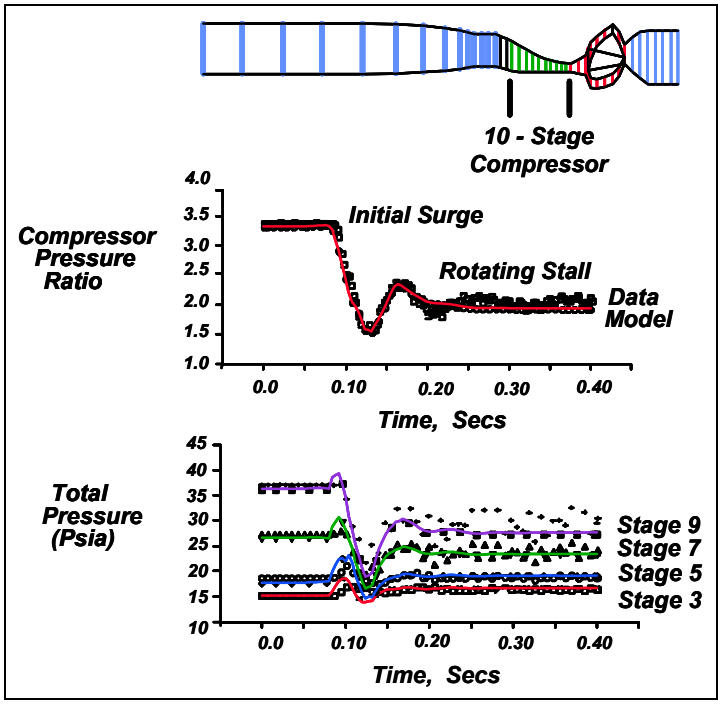


Figure A.10: Post-Stall Dynamic Event – DYNTECC.

Limitations of Chosen Modeling Technique

For the stage-by-stage compression system modeling technique chosen, the biggest issue is the development of stage characteristics. Generally, even compressor rig tests are not instrumented well enough to get proper stage characteristic information. If blade shapes are known, a mean-line or streamline curvature technique can be used to generate characteristics. However, these types of codes rely on empirical inputs based upon cascade test results and generic correlations. In addition the DYNTECC code, even though the single spool version has a parallel compressor capability, does not handle radial distortion well and the dual-spool version of DYNTECC is currently not configured for parallel compressors.

A.3.1.2 Engine Stall Using One-Dimensional Modeling

The gas turbine engine has played a significant role in the advancement of the flight capabilities of modern day aircraft. In order for a gas turbine engine to operate at the performance, operability, and durability level for which it was designed, stable operation of the various engine components must be ensured. Transient and dynamic instabilities, which could push the engine components beyond their operational limits, could result in loss of thrust, loss of engine control, or possible engine damage due to high heat loads and high cyclic stresses. The influence of operating instabilities must be quantified not only from the individual component considerations, but also from the point of view of any interaction between the various components.

Modeling Techniques Used

The turbine engine modeling technique was the Aerodynamic Turbine Engine Code, or ATEC, [A.4 and A.5], a time-dependent turbine engine model and simulation capable of simulating a turbojet engine operating in both transient and dynamic modes. Other gas turbine engine models and simulations have typically focused on providing a transient, component-level representation of the overall engine, or a dynamic representation of a single component. The ATEC simulation provides a bridge between the two types of simulations. It provides the computational efficiency that is desired when simulating the gas turbine engine during transient events, but it also provides the appropriate simulation techniques to address overall engine operation during a dynamic event such as compressor surge or combustor blow-out. ATEC provides the detailed system resolution needed to analyze a dynamic event (such as a stage-by-stage representation of the compression system), but uses the same type of component performance information used in standard transient simulations.

The Aerodynamic Turbine Engine Code (ATEC) solves the one-dimensional, time dependent, compressible, inviscid flow-field solution for internal flows by solving the Euler equations for the conservation of mass, momentum, and energy. The solution is obtained over the computational domain using both implicit and explicit numerical integration routines. The effects of the various engine system components are modeled using turbomachinery source terms in the governing equations.

Potential Benefits

The ATEC model and simulation can simulate on and off-design steady-state operation, as well as transient and dynamic engine responses to perturbations in a wide range of operational and control conditions. By example, it has been shown that the ATEC simulation can handle a wide variety of conditions that occur during normal and abnormal gas turbine engine operation. The benefits of the variable time-step routine, which uses a combination of the explicit and implicit numerical solvers, were also demonstrated. Test cases were presented that demonstrated that the ATEC simulation could: be calibrated to a steady-state data set; extended the steady-state calibration to a transient fuel variation; presented results from an engine operation that resulted in compressor surge; and addressed a turboshaft engine going through the start process. The ATEC results were shown to agree closely with test data where available.

Cited Examples

- Garrard, G.D., "ATEC: The Aerodynamic Turbine Engine Code for the Analysis of Transient and Dynamic Gas Turbine Engine System Operations – Part 1: Model Development", ASME Paper # 96-GT-193, June 1996. **[A.4]**
- Garrard, G.D., "ATEC: The Aerodynamic Turbine Engine Code for the Analysis of Transient and Dynamic Gas Turbine Engine System Operations – Part 2: Numerical Simulations", ASME Paper # 96-GT-194, June 1996. **[A.5]**

This example demonstrates the real benefit of a dynamic simulation, by computing post-stall operation of a compressor and engine system, which cannot be modeled with a cycle-type simulation. The test case simulated a transient throttle movement using the T55-L-712 that resulted in the gas generator portion of the engine decelerating from 100 percent speed to approximately 90 percent speed. After a brief pause at the 90 percent speed, the engine was accelerated back to the 100 percent speed condition. The change in fuel flow rate during the acceleration was fast enough to force the compressor into surge cycles. The relative compressor pressure ratio as a function of time is illustrated in the figure below. The multiple surge cycles cause a significant drop in the total pressure throughout the engine and a corresponding flow reversal as indicated in **Figure A.11**.

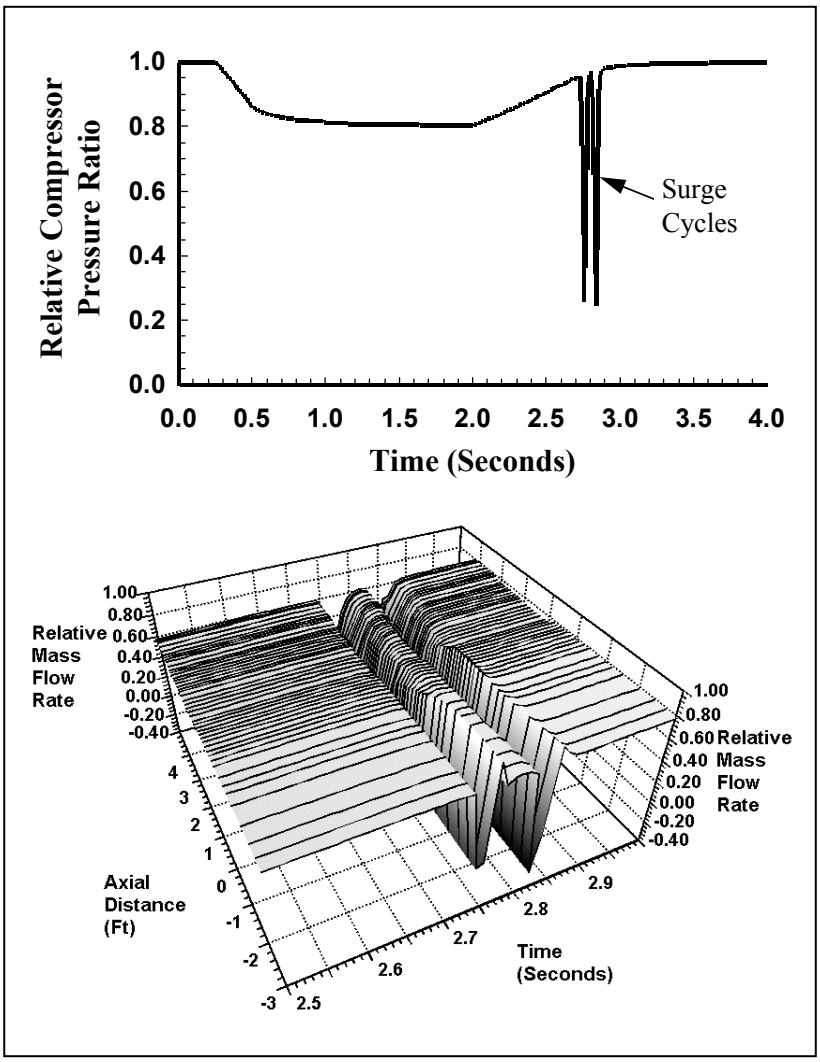


Figure A.11: Surge Cycles during Engine Re-Acceleration.

Limitations of Chosen Modeling Technique

The simulation described in the cited reference uses a one-dimensional approach. Thus, the interaction associated with inlet spatial distortion can not be analyzed with this modeling technique. It is, however, an excellent technique to investigate control actions that can be initiated because of some destabilizing event such as an inlet by-pass door malfunction or an engine surge.

A.3.1.3 Combustor Dynamics

Due to the inherent dynamics of the fluid system, transient performance of a gas turbine engine can differ significantly from that predicted from quasi-steady operating assumptions. The consequences of these can be quite dramatic, including unexpected crossing of the compressor surge line while changing operating points. Beyond the surge line, compressor rotating stall and surge serve as the forcing function for a complex dynamic interaction between engine components. Because these unsteady operating cycles produce substantially reduced performance and durability the recovery from the instability is an important issue facing the gas turbine designer. Because of this, significant efforts have been made to accurately simulate the performance of a compressor undergoing a surge transient. As these techniques have matured, the focus increasingly has shifted to an extension of these methods to encompass the entire engine, and thus capture the important compressor-combustor interactions that occur during engine surge.

Modeling Techniques Used

Design methods for gas turbine engine combustors require mathematical models that satisfy two simultaneous and often-conflicting requirements. These are to provide an accurate description of the highly complex geometry and physics involved, and be sufficiently inexpensive in computational requirements to allow its incorporation in a design cycle involving the evaluation of a great number of operating conditions. For these reasons, a one-dimensional, finite-rate, unsteady combustor model has been developed that incorporates most elements found in modern gas turbine burners, and yet is simple enough to be implemented in desktop computers.

Combustor models for dynamic behavior may have a division of the flow path into annular and primary streams with finite-rate effects within the primary flow and interaction between hot and cold gases through dilution holes. Over the past 50 years there have been numerous efforts at modeling gas-turbine combustor performance. Physical models and numerical techniques have primarily been developed for the simulation of steady flow in combustors. In the past 20 years, particular emphasis has been placed on engine pollution. One of the key difficulties in modeling dynamic engine behavior has been the limited knowledge of transient combustion phenomena, or more specifically post-stall combustor dynamics. Initial efforts employed a quasi-steady heat release formulation based on the fuel-air ratio and only simple combustion efficiency degradation was modeled. Two recent modeling efforts have made improvements based on finite-rate chemistry for incorporation into a dynamic gas turbine engine model.

Potential Benefits

Dynamic modeling of combustor and compressor interaction can lead to a better understanding of the relationship between combustor blowout and re-light during engine surge. When a multi-zoned model is used, combustor flow within the primary zone can be modeled with finite-rate chemistry to allow predictions of blowout and the effects of perturbation in boundary and operating conditions.

Cited Examples

- Rodriguez, C.G. and O'Brien, W.F., "Unsteady, Finite-Rate Model for Application in the Design of Complete Gas-Turbine Combustor Configurations", Design Principles and Methods for Aircraft Gas Turbine Engines, RTO-MP-8, February 1999. **[A.1]**

- Costura, D.M. et al., "A Model for Combustor Dynamics for Inclusion in a Dynamic Gas Turbine Simulation Code", AIAA Paper # 97-3336, Presented at the 33rd AIAA/ASME/SAE/ASEE Joint Propulsion Conference, July 6-9, 1997, Seattle, WA. [A.13]

Indicated in **Figure A.12**, is a solution grid for a generic reverse flow combustor used in an analysis as presented in **Ref. [A.1]**. The main issues associated with the layout of the grid from the point of view of a one-dimensional theory are division of the main flow into primary and annular paths, and interaction between flow paths through the presence of dilution holes. Once the grid has been selected, analysis can be conducted to determine the effects of perturbations in boundary and operating conditions. Indicated in **Figure A.13**, are the effects of an imposed fuel flow oscillation on total pressure.

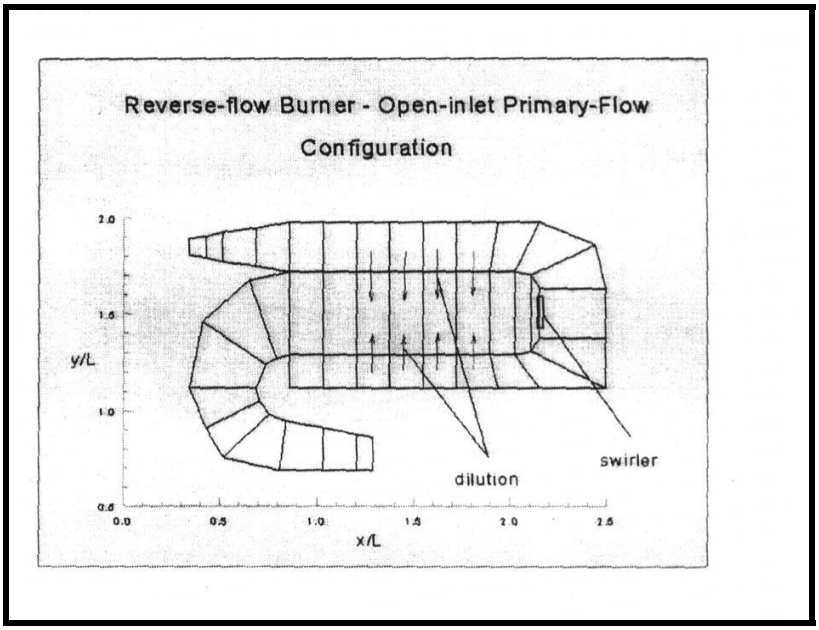
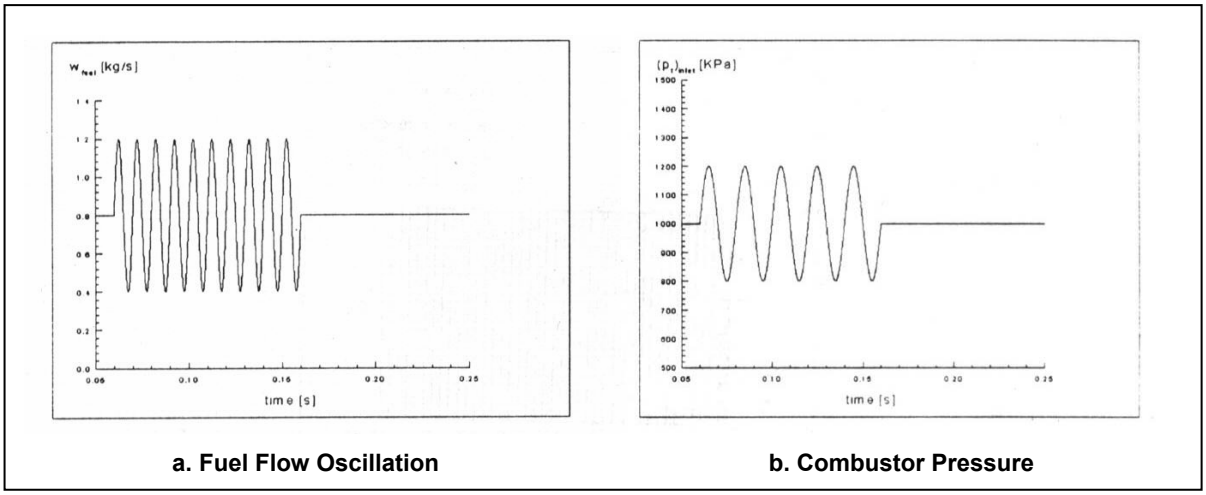


Figure A.12: Grid of Reverse Flow Burner.



a. Fuel Flow Oscillation **b. Combustor Pressure**

Figure A.13: Effects of an Imposed Fuel Flow Oscillation on Combustor Pressure.

Combustor performance for a dump combustor as illustrated in **Figure A.14** during ignition and blowout are illustrated in **Figure A.15** and **Figure A.16**, taken from **Ref. [A.13]**. During the ignition event, airflow

unsteadiness was quite high during the period of ignition from t = 6 to t = 7 seconds. An oscillation of 8 psi in magnitude and 50 Hz frequency is seen, indicative of an initial instability in the combustor. For a lean blowout case, a gradual fuel reduction began at 7 seconds with complete fuel shut-off achieved near t = 13 seconds. Note the minimum fuel flow that can be recorded is 0.5 gph, and thus the exact point of fuel extinction is expected between t = 13 and t = 14 seconds.

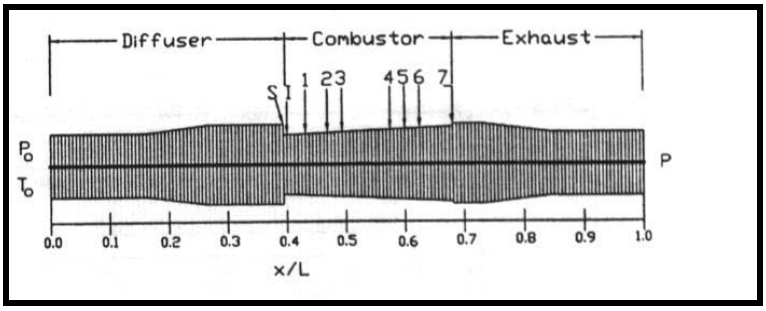


Figure A.14: Grid for Dump Combustor.

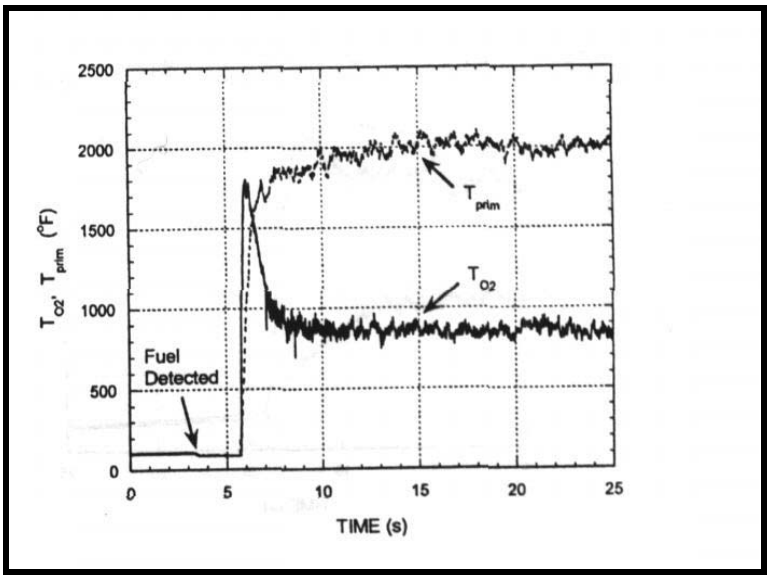


Figure A.15: Ignition Event.

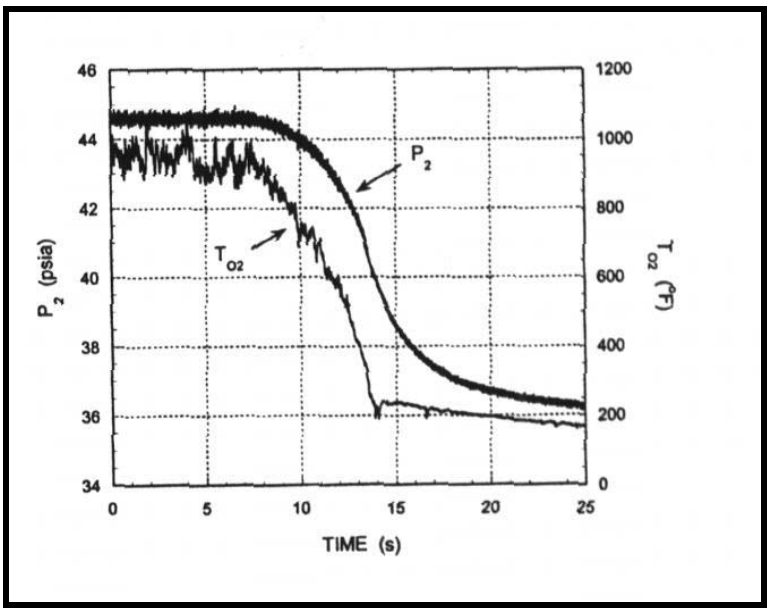


Figure A.16: Blowout Event.

Limitations of Chosen Modeling Technique

Unsteady models, based on the fundamental governing equations for flows with non-equilibrium chemistry, have been developed for gas turbine configurations. The chosen approach in both models is a one-dimensional fluid dynamics approach with integral conservation equations for multi-species flows with chemical reactions. All the effects that could not be handled by the usual one-dimensional ideal flow (Euler) equations (i.e. area changes and friction) were included in source terms on the right-hand side of the system. In **Ref. [A.1]**, a one-step chemistry model was used and in **Ref. [A.13]**, a two step model was used. Both models can provide steady state and dynamic results. The results from these models were satisfactory, but showed some limitation of any one-dimensional approach in that pressure losses could not be modeled by simple fluid dynamics, and were provided to the model.

A.3.1.4 Engine-Inlet Integration

The economic viability of a commercial supersonic transport, such as the High Speed Civil Transport (HSCT), is highly dependent on the development of a high-performance propulsion system. Typically, these propulsion systems mate a supersonic mixed-compression inlet with a turbojet or turbofan engine. The nature of such propulsion systems offers the potential for undesirable component interactions, which must be thoroughly understood for proper design. Therefore, it is imperative to have tools that allow investigation of inlet-engine integration issues.

The inlet must provide the engine with the correct mass flow rate at the highest possible pressure with minimum drag. Additionally, flow angularity and distortion must be minimized at the compressor face if the engine is to function appropriately. Maximum thrust with a minimum of fuel consumption will not be obtained without the inlet operating close to peak performance. Unfortunately, operating near peak performance can result in an inlet unstart (expulsion of the normal shock) followed by engine stall and possibly surge. When that happens, proper control action must be taken to recover the system as quickly as possible. Thus the operability of the overall system must also be addressed, because stable time dependent operation of the system must be ensured for both scheduled and non-scheduled events.

Because of the complexity of the inlet and engine systems, and the high cost of experimentally determining overall performance, numerical simulations of the components can be of significant benefit.

For example, dynamic simulations provide a means for investigating the potential interactions mentioned above, as well as providing a test bed for guiding the design, testing and validation of propulsion controls.

Modeling Techniques Used

The simulation system, operating under the Application Portable Parallel Library (APPL), closely coupled a supersonic inlet with a gas turbine engine. The supersonic inlet was modeled using the Large Perturbation Inlet (LAPIN) computer code, and the gas turbine engine was modeled using the Aerodynamic Turbine Engine Code (ATEC) as illustrated in **Figure A.17**.

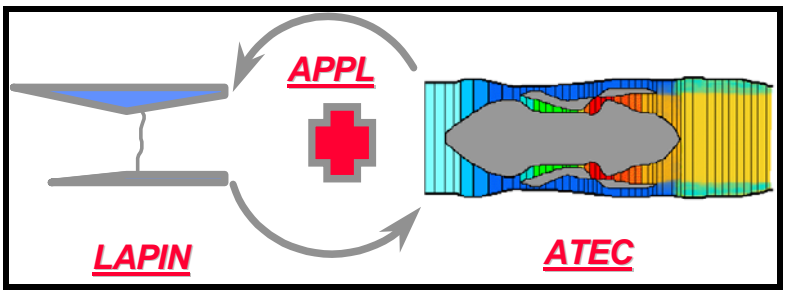


Figure A.17: Schematic of the Coupled Inlet-Engine Codes.

Both LAPIN and ATEC provide a one-dimensional, compressible, time dependent flow solution by solving the one-dimensional Euler equations for the conservation of mass, momentum, and energy. Source terms are used to model features such as bleed flows, turbomachinery component characteristics, and inlet subsonic spillage while unstarted. High frequency events, such as compressor surge and inlet unstart, can be simulated with a high degree of fidelity. The simulation system was exercised using a supersonic inlet with sixty percent of the supersonic area contraction occurring internally, and a GE J85-13 turbojet engine as illustrated in **Figure A.18**.

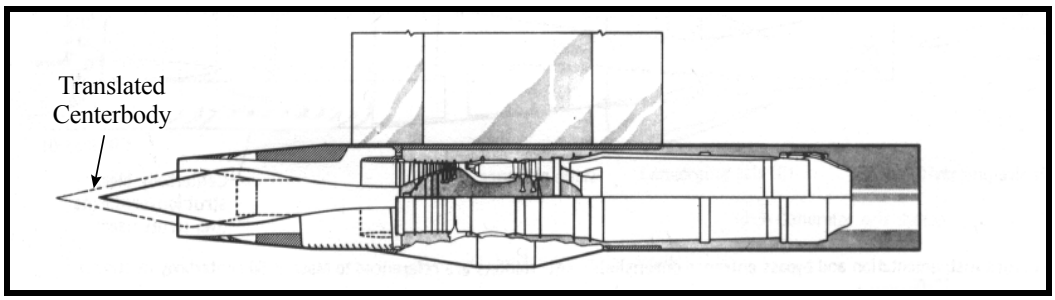


Figure A.18: Cross-Section View of the 4060 Inlet and J85-13 Turbojet Installation.

The cited paper **[A.14]** describes the general modeling techniques used in the simulations, and the approach taken to implement the simulations under the APPL environment. It presents results from selected test cases.

Potential Benefits

Traditionally, aircraft inlet performance and propulsion performance have been designed separately and latter mated together via flight-testing. In today's atmosphere of declining resources, it is imperative that more productive ways of designing and verifying aircraft and propulsion performance be made available to the aerospace industry. One method of obtaining a more productive design and evaluation capability is

with numerical simulations. Numerical simulations can provide insight into physical phenomena that may not be understood by test data alone. Simulations can fill information gaps and extend the range of test results to areas not tested. In addition, once a simulation has been validated, it can become a numerical experiment and the analysis engineer can conduct 'what-if' studies to determine possible solutions to performance or operability problems.

Cited Example

- Garrard, G.D., Davis, Jr., M.W., Wehofer, S. and Cole, G., "A One-Dimensional, Time Dependent Inlet/Engine Numerical Simulation for Aircraft Propulsion Systems", ASME Paper # 97-GT-333, June 1997. **[A.14]**

The simulation system was exercised using a supersonic inlet with sixty percent of the supersonic area contraction occurring internally, and a GE J85-13 turbojet engine. The inlet-engine simulation combination of LAPIN and ATEC was compared to experimental results. A transient event was initiated at a flight Mach number of 2.5 by pulsing the bypass doors in the closed direction. The result was an inlet unstart followed by an engine compression system stall. During the given transient, the majority of the system instabilities can be traced to the fact that the normal shock, located initially downstream of the inlet throat, was expelled outside of the inlet. The location of the shock is plotted as a function of time in **Figure A.19**. The shock location is normalized by the inlet cowl lip radius, and referenced to the centerbody tip. The cowl lip is axially located two cowl-lip-radii downstream of the centerbody tip. The act of closing the bypass valve forces the shock structure to be expelled from the inlet. Moving the centerbody forward in conjunction with proper modulation of the bypass doors allows the shock to be re-ingested.

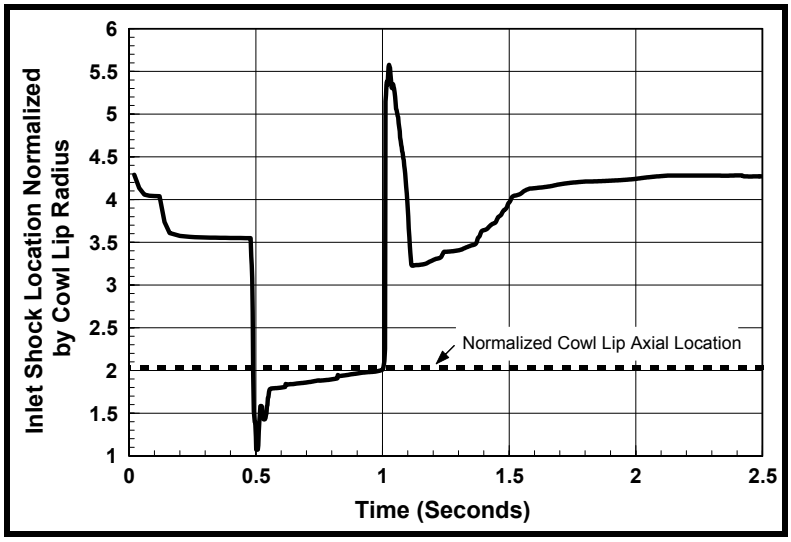


Figure A.19: Inlet Shock Location.

Total pressure across the compressor is lost once the inlet system unstarts. At the instant of unstart there is a sharp spike in pressure ratio to a value exceeding 5.0 which (probably) exceeds the steady-state stall line, resulting in stall. Although the system begins to recover the original level of compressor operating pressure ratio, the compressor total pressure ratio is lower. The relative compressor pressure ratio is plotted as a function of the inlet corrected mass flow rate, expressed as a percentage of the design mass flow rate, in **Figure A.20**. It is evident from the figure that there is one engine surge cycle, with a rotating stall event. Recovery takes place at a lower corrected inlet mass flow rate due to the lower engine shaft speed.

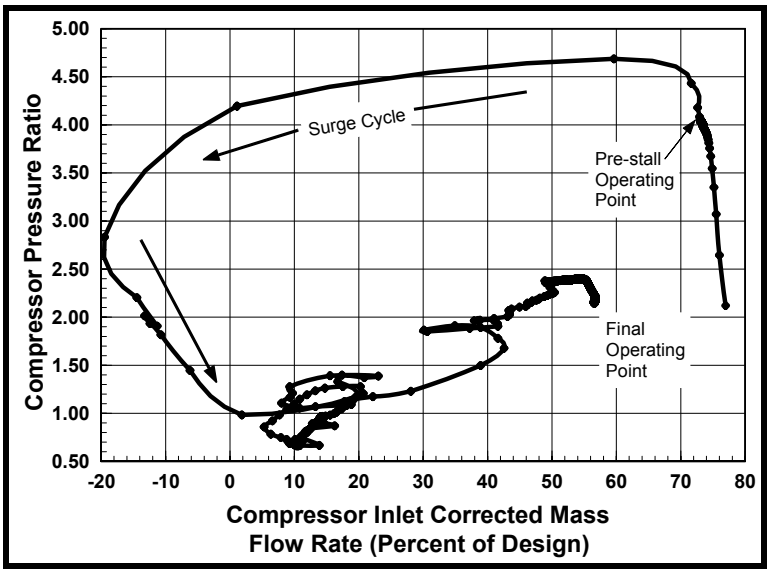


Figure A.20: Compressor Pressure Ratio as a Function of Compressor Inlet Corrected Mass Flow Rate.

Limitations of Chosen Modeling Technique

The simulation described in the cited reference uses a one-dimensional approach. Thus, the interaction associated with inlet spatial distortion cannot be analyzed with this modeling technique. It is, however, an excellent technique to investigate control actions that can be initiated by some destabilizing event such as an inlet by-pass door malfunction or an engine surge.

A.3.1.5 Hot Gas Ingestion

Engine stability problems have occurred when exhaust products from rocket or gun firings are ingested into the engine. The most common form of instability is axial compressor stall which, depending on engine type and the aircraft flight envelop, can lead to surge with serious and damaging effects. From previous investigations, it has been determined that the rapid inlet temperature increase is probably the predominant factor causing compression system instability in these situations. Tests with rapid inlet temperature transient rates in the range of 2000 – 8000 K/sec are typical of those experienced during gun or rocket gas ingestion.

Modeling Techniques Used

The numerical simulation used for this type of application was a stage-by-stage compression system model known as *DYNTECC* (*DYN*amic *T*urbine *E*ngine *C*ompressor *C*ode) [A.2]. DYNTECC is able to analyze post-stall behavior and predict the onset of compression system instability. Stability limit analysis can be conducted for single-spool and dual-spool systems with and without temperature or pressure distortion. DYNTECC requires a full set of stage characteristics and annulus geometry as inputs. DYNTECC was configured and applied to the geometry of the TF30 compression system as illustrated in **Figure A.21**, for an investigation into the effects of temperature ramps on system stability. The TF30 compression system has dual-spool rotors, but with three compressors: fan, low-pressure and high-pressure compressors.

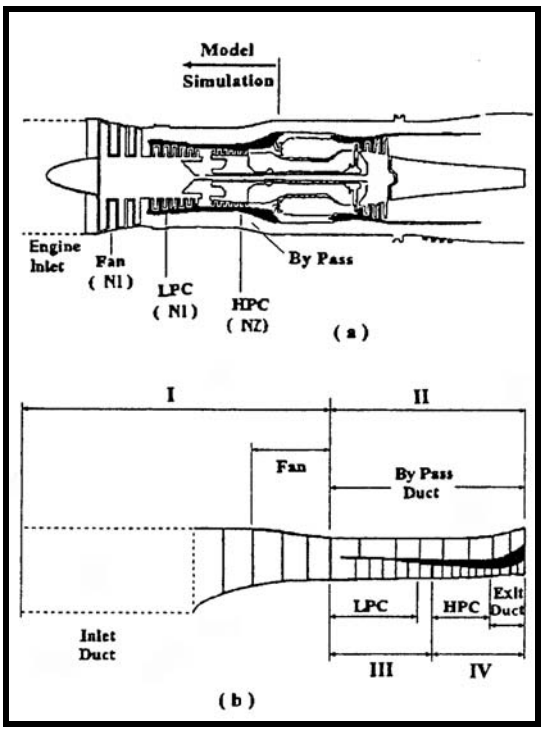


Figure A.21: Schematic of TF30 Compression System Modeled.

Potential Benefits

Mathematical compressor models have become a major tool for understanding compression system behavior during dynamic events such as inlet distortions. A validated compression system model can be used to extend the range of the experimental test results to the untested regime. Using a validated stage-by-stage compression system model, a parametric investigation can determine the qualitative effects of gas ingestion effects on system operability.

Cited Example

- Abdel-Fattah, A.M., "Response of a Turbofan Engine Compression System to Disturbed Inlet Conditions", Journal of Turbomachinery, Vol. 119, No. 4, October 1997. **[A.15]**

Using the DYNTECC stage-by-stage compression system simulation, the following observations were made with regard to the TF30 gas ingestion investigation:

- The stability limit of the system in response to inlet temperature ramps improved with increasing low rotor speed.
- The stability limit, in terms of temperature rise required to surge the compressor, was found to be independent of the rise rate as illustrated in **Figure A.23**.
- At the time of instability because of inlet temperature ramps, the DYNTECC model predicted the possibility of the third stage of the fan as the critical stage responsible for compression system surge initiation as illustrated in **Figure A.22**.

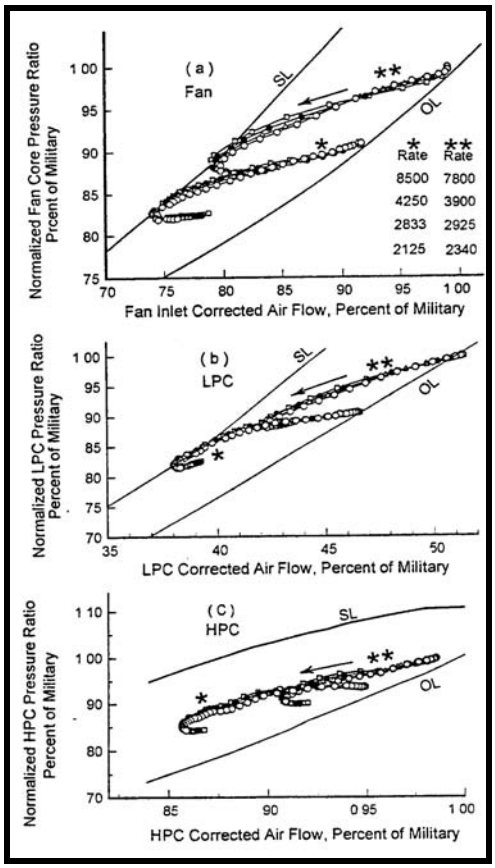


Figure A.22: Ops Line Migration due to Temperature Ramps.

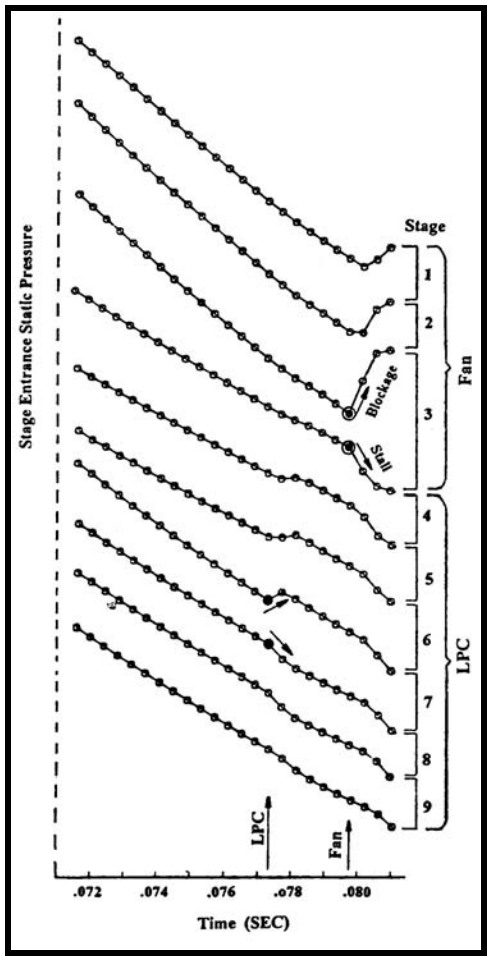


Figure A.23: Static Pressure Signature.

Limitations of Chosen Modeling Technique

For the stage-by-stage compression system modeling technique chosen, the biggest issue is the development of stage characteristics. Generally, even compressor rig tests are not instrumented well enough to get proper stage characteristic information. If blade shapes are known, a mean-line or streamline curvature technique can be used to generate characteristics. However, these types of codes rely on empirical inputs based upon cascade test results and generic correlation. In addition, the DYNTECC code does not handle radial distortion well, even though the single spool version has a parallel compressor capability. The dual-spool version of DYNTECC is currently not configured for parallel compressors.

A.3.1.6 Distortion Investigation Using Parallel Compressor Model

Modern high-performance military aircraft are subjected to rapid flight maneuvers, which place great operational demands on their air-breathing gas turbine engines. One component of the engine that is particularly sensitive to the fluid dynamic transients that result from rapid aircraft maneuvers is the compressor. The compressor should operate in a stable manner during all aspects of flight. However, rapid flight transients cause the aircraft intake to present a highly distorted total pressure flow-field to the compressor inlet. High distortion levels may cause the compressor to surge at high rotational speeds or slip into rotating stall at lower rotational speeds. Since total pressure distortion is the primary reason

for reaching the engine stability limit, its effects on system performance and operability need to be understood.

Modeling Techniques Used

For this investigation, a one-dimensional compression system model with a parallel compression system modeling technique was used (DYNTECC). Parallel compressor theory states that the overall control volume representing the compressor and inlet or exit duct system may be circumferentially divided into segments. Each segment then acts in parallel with each other segment, exiting to the same exit boundary condition. Different magnitudes of inlet total pressure and temperature can then be imposed upon each segment of the parallel compressor as illustrated in **Figure A.24**. In the purest sense, each segment is independent of all other segments, except through the exit boundary condition.

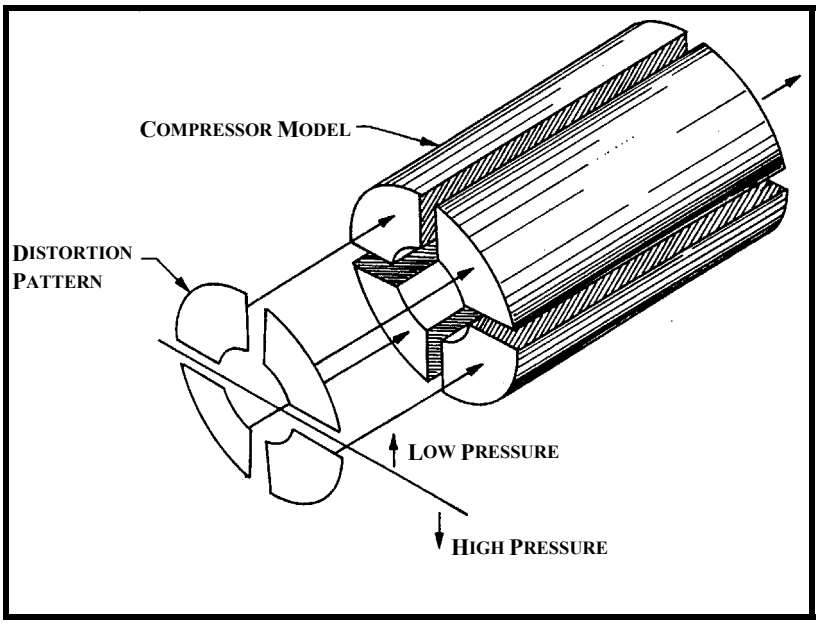


Figure A.24: Parallel Compressor Theory.

Each circumferential segment was modeled using a one-dimensional approach with an overall control volume for each. Each overall control volume was then subdivided into elemental control volumes. The governing equations were derived by the application of mass, momentum, and energy conservation principles to the elemental control volume. In the compressor section, a stage elemental control volume consists of a rotor followed by a stator and associated volume representing the complete stage. Acting on this fluid control volume is an axial-force distribution, FX, attributable to the effects of the compressor blading and walls of the system. In addition, the rate of heat transfer to the fluid and shaft work done on the fluid are represented by distributions, Q and SW, respectively.

Potential Benefits

Mathematical compressor models have become a major tool for understanding compression system behavior during dynamic events such as inlet distortions. A validated compression system model can be used to extend the range of the experimental test results to the untested regime. Using a validated parallel compressor model, a parametric investigation can be conducted to determine the qualitative effects of certain combinations of transient and steady state pressure and temperature distortions on system operability.

Cited Example

- Davis, Jr., M.W. et al., "Euler Modeling Techniques for the Investigation of Unsteady Dynamic Compression System Behavior", Loss Mechanisms and Unsteady Flows in Turbomachines, AGARD-CP-571, January 1996. [A.3]

DYNTECC can be configured to analyze inlet total pressure distortion using a modified parallel compressor theory. DYNTECC was used to analyze the effects of inlet distortion on a two-stage, low aspect ratio fan. The model was initially calibrated against the experimental clean inlet performance of the compressor. The modified model was then validated using the compressor performance with a pure circumferential inlet distortion pattern. The inlet distortion pattern was generated from a 1/rev circumferential distortion screen shown in **Figure A.25**.

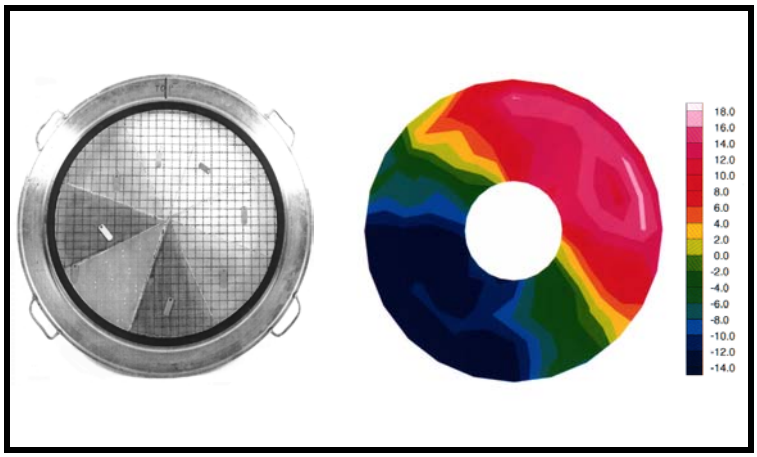


Figure A.25: Imposed Distortion Pattern, Produced by Distortion Screen.

This screen produces a total pressure inlet distortion that is purely circumferential. The simulation was run with the model modifications, as well as with the pure parallel compressor theory for comparison purposes. Shown in **Figure A.26** are the model predictions, compared to the experimental results at 98.6% speed. The inlet distortion pattern has a significant impact on both compressor performance and operability.

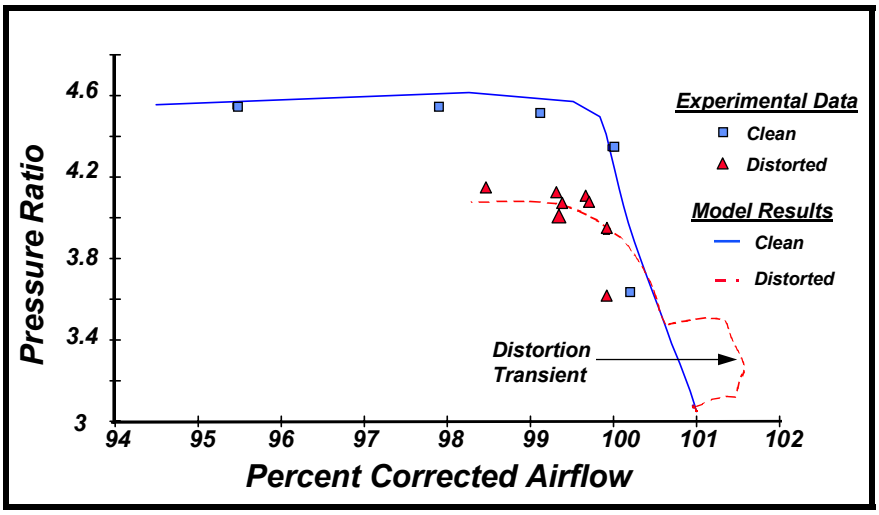


Figure A.26: Model Prediction and Comparison.

Limitations of Chosen Modeling Technique

For the parallel compressor theory, system instability occurs when any one segment becomes unstable as a result of the inlet and exit conditions imposed upon it. This assumption has been successful when the simulation is limited to few segments. The assumption becomes invalid when the simulation uses many circumferential segments without a mechanism to redistribute circumferential flow. Some simulations have improved upon this concept by allowing circumferential cross-flow or radial redistribution. However, these improvements are usually empirical in nature and rely upon specific databases.

A.3.2 Three-Dimensional Model Applications

The following applications use modeling techniques that range from solving the 3-D Euler equations to full 3-D Navier-Stokes equations. Each has their advantages and disadvantages and contributes in specific areas. No one 3-D modeling technique can solve all types of problems and as such each is suited to a specific type of application. The reader is encouraged to investigate the references since what is presented in this section is a summary of each reference.

A.3.2.1 Distortion Investigation

Modern high-performance military aircraft are subjected to rapid flight maneuvers, which place great operational demands on their air-breathing gas turbine engines. One component of the engine that is particularly sensitive to the fluid dynamic transients that result from rapid aircraft maneuvers is the compressor. The compressor should operate in a stable manner during all aspects of flight. However, rapid flight transients cause the inlet to produce a highly distorted total pressure flow-field to the compressor inlet. High distortion levels may cause the compressor to surge at high rotational speeds or slip into rotating stall at lower rotational speeds. Since total pressure distortion may cause the compression system to reach the stability limit earlier, its effects on system performance and operability need to be understood. Distortion imposed on a circumferentially swirling flow was shown by Greitzer and Strand (1978) to have a three-dimensional (3-D) nature, which is fundamental to the development of both inlets and compressors. Design or analysis engineers are interested in understanding the details of the flow-field to determine the effects of inlet total pressure distortion on the compressor. One way to quantify the effects of distortion is to test for that effect in a ground test facility. Another way is to analyze the flow-field using a computational technique.

Modeling Techniques Used

Parallel compressor theory has been used successfully in the past to develop an understanding of compressor performance with inlet distortion. However, parallel compressor theory has been generally restricted to simple inlet distortion patterns and is generally conservative at estimating the stability limit. Investigators have made extensive modifications to parallel compressor theory through modeling techniques to account for the transfer of mass, momentum, and energy transfer between segments. These techniques are empirically based and have to be applied on a case-by-case basis. Recent developments of three-dimensional simulations can automatically account for the transfer of conservation properties.

A 3-D numerical simulation for analysis of inlet distortion has been developed and is known as the Turbine Engine Analysis Compressor Code (TEACC) **[A.9]**. TEACC solves the compressible three-dimensional (3-D) Euler equations over a finite-volume grid domain through each blade row. The Euler equations are modified to include turbomachinery source terms of bleed flow, blade forces (in the three Cartesian directions), and shaft work, which model the effect of the blades. The source terms are calculated for each circumferential grid section of each blade row by the application of a streamline curvature code. A methodology was developed for distributing the turbomachinery source terms axially, radially, and circumferentially through the bladed region. The overall TEACC development methodology is conceptually presented in **Figure A.27**.

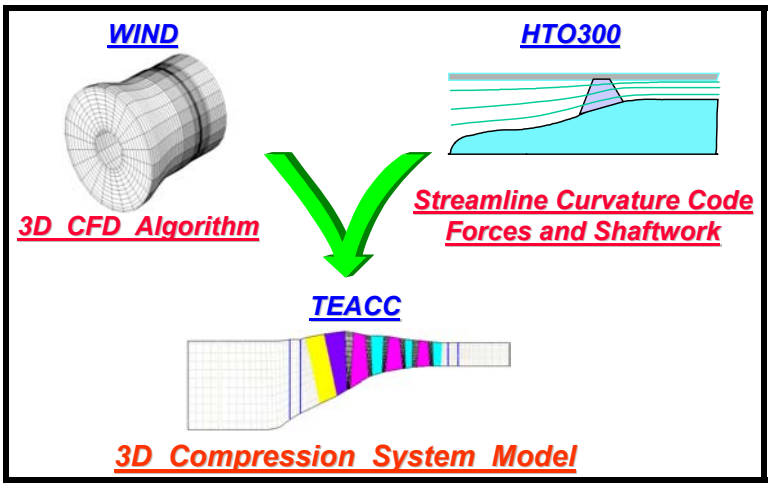


Figure A.27: General Technical Approach for 3-D Dynamic Compression System Model, TEACC.

Potential Benefits

Distortion imposed on a circumferentially swirling flow has been shown to be three-dimensional (3-D) in nature. Design or analysis engineers are interested in understanding the details of the flow-field to determine the effects of inlet total pressure distortion on the compressor. One way to quantify the effects of distortion is to test for that effect in a ground test facility. Currently, the inlet and engine are tested separately. Typically, the aircraft fuselage is too big to fit in a wind tunnel. A forebody simulator is used in conjunction with the inlet to characterize its flow-field. The forebody simulator is designed to produce a flow-field at the inlet reference plane (IRP) similar to the flow-field produced by the aircraft. Screens are constructed to capture the most severe dynamic patterns produced by the inlet and are then placed in front of the engine to measure the loss of stall margin produced by the steady-state inlet distortion. However, it is expensive to instrument a compressor and perform the necessary number of tests to adequately understand the compressor flow-field. Numerical simulations have been developed to support the testing community in this area.

Cited Examples

- Hale, A.A. and O'Brien, W.F., "A Three-Dimensional Turbine Engine Analysis Compressor Code (TEACC) for Steady-State Inlet Distortion", Journal of Turbomachinery, Vol. 120, July 1998, pp. 422-430. **[A.9]**
- Hale, A.A., Chalk, J., Klepper, J. and Kneile, K., "Turbine Engine Analysis Compressor Code: TEACC – Part II: Multi-Stage Compressors and Inlet Distortion", AIAA Paper # 99-3214, Presented at the 17th AIAA Applied Aerodynamics Conference, June 1999. **[A.16]**

A three-stage fan, representing a modern, high-performance military fan was modeled. It consists of a structural strut, a variable inlet guide vane (IGV) attached to the back of the strut, and three rotor-stator pairs. Only overall experimental performance data were available for comparison. No blade-row by blade-row or radial distributions of flow quantities were available. Consequently, only overall performance was compared with data. An axial-radial representation of the grid used for the three-stage fan is shown in **Figure A.28**. The three-dimensional grid on which the flow-field solution was obtained was uniformly spaced in the circumferential direction at 15-degree intervals.

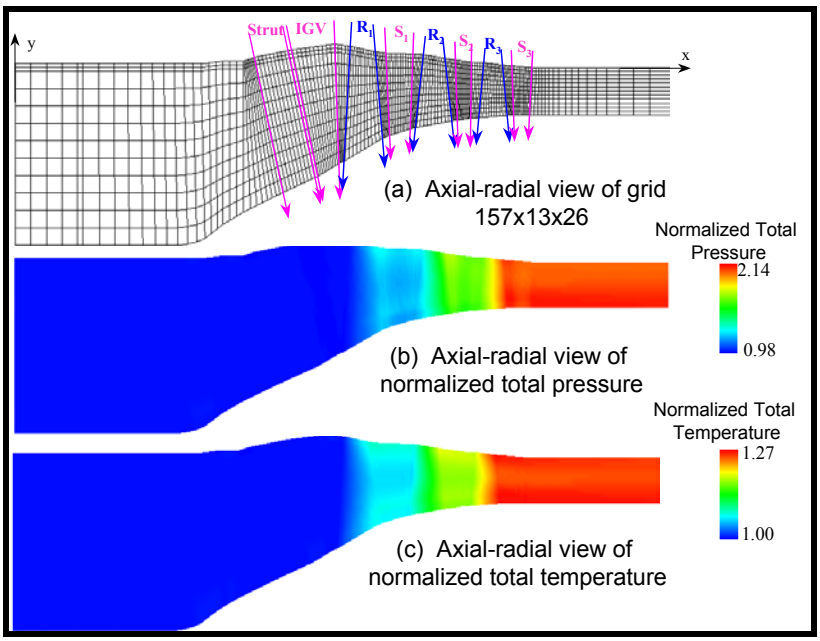


Figure A.28: Grid and Performance for Three-Stage Fan at 80-percent Corrected Speed.

The fan was executed with clean, standard-day inlet conditions. The results of the TEACC fan simulation are at 68% of the design corrected mass flow on the 80% speed line. Both the normalized pressure ratio and the normalized temperature ratio achieved at this point were within 0.5% of the experimental data. These results were achieved with uncalibrated loss and deviation correlations obtained from the open literature. As expected, there is an increase in both total pressure and total temperature through the machine. **Figure A.28** provides an insight into the general character of how the total pressure and temperature vary radially and axially throughout the machine.

Illustrated in **Figure A.29** is the calibrated clean inlet performance of the three-stage fan at various speeds and a corresponding flow-field for one particular point on the map. Illustrated in the flow-field view are streamlines, colored with Mach number, through the fan. This figure clearly illustrates the rotors turning the flow and the stators straightening the flow. The total turning of the flow through the entire machine is about 45 degrees. The hub has the highest Mach number through the entire fan. The streamlines illustrate that the flow is almost completely axial at the exit of the fan.

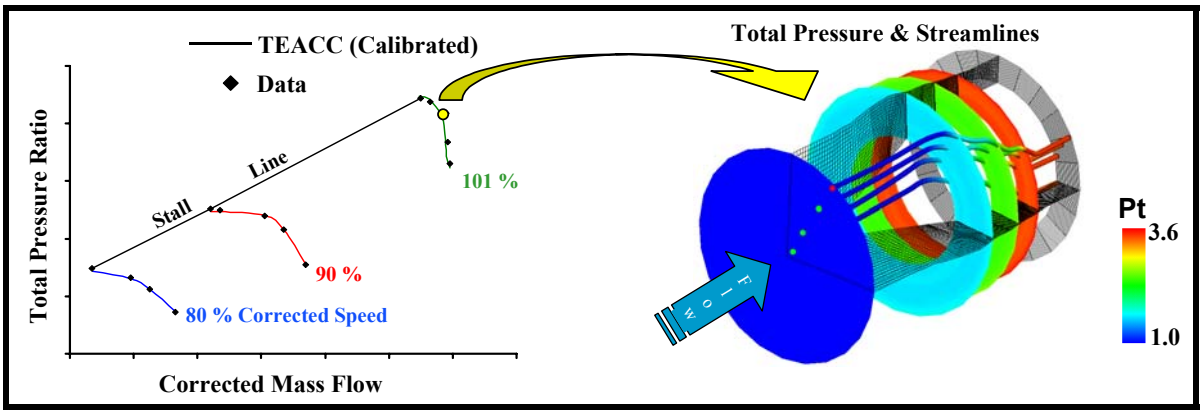


Figure A.29: Point Inlet Performance and Corresponding Flow-Field for a Typical Three-Stage Military Fan.

This same three-stage fan was exercised with a total pressure distortion flow-field imposed upon its inlet as illustrated in **Figure A.30**. The distortion was simulated with as a steady state pattern produced by an inlet distortion screen, a standard ground test method for simulating distortion at the engine inlet plane. Although, the model results are not compared to experimental results there is enough information to make an engineering judgment as to the validity of the results.

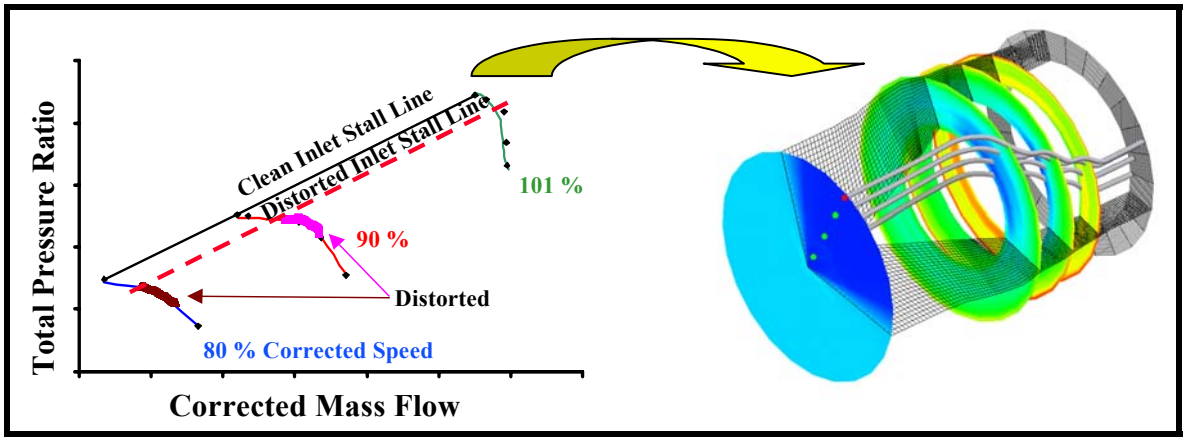


Figure A.30: Predicted Performance with Imposed 90° Circumferential Distortion.

Limitations of Chosen Modeling Technique

This modeling technique has its strengths and weakness. The approach allows for a relatively quick computation of distortion when compared to traditional CFD turbomachinery, but because there is a level of empiricism the flow-field may not be accurately represented at a detailed level suitable for design. This modeling technique is best suited for studying potential design changes that can be quantified as to their effects on blade performance and system response to inlet distortion.

A.3.2.2 Rotating Stall and Surge Investigation

Compressor instability is a major limiting factor on gas turbine engine operating range, performance, and reliability. The instability, either in a form known as rotating stall or surge, occurs at an operating point with low mass-flow and a large pressure rise. To avoid such instabilities, the compressor has to run at an operating point corresponding to a lower pressure ratio so that an adequate stall margin is maintained. The stall margin can be badly reduced in operating environments for which the inlet conditions are non-uniform. Predicting the condition at which instability will occur in a compressor requires an understanding of the flow process leading to the onset of the instability. The transition from initial disturbance to final stall or surge can usefully be divided into three stages: (1) inception, (2) development, and (3) final flow pattern. The inception stage is the period when disturbances start to grow (flow becomes unstable). It defines the operating point and conditions at which instability occurs. In practice, the disturbances will take a finite amount of time, ranging from a few to several hundred rotor-revolutions, to grow into final stall or surge. So, the inception stage can be viewed as the early development of the unstable flow. The development stage, which includes all the processes after the inception stage through to the final flow pattern, is usually of less importance. It is often the case that one final form of instability in one compressor could be the pre-stage of the final form in another compressor.

Modeling Techniques Used

A non-linear three-dimensional computational model aimed at simulating 3-D finite amplitude disturbances such as inlet distortions, short wavelength stall inception processes, and part-span stall cells

has been developed as outlined in the cited reference [A.17]. To make the model practicable in terms of currently available computational resource, the following simplifications were made:

- Infinite number of blades assumption – the resolution of flow-field in every blade passage is not computationally feasible with currently available computational resources.
- A local pressure rise characteristic in every small portion of a blade passage can be defined. It is essential for a blade row to respond in a local manner, since flow redistribution is expected within a blade row. This assumption is consistent with the infinite number of blade assumption and is thus good for a blade passage of high solidity.

Since the number of blades is infinite in a bladed region, the flow at each circumferential position can be regarded as axisymmetric flow in a coordinate frame that is fixed to the blade row. The pressure rise and flow turning due to blades can thus be simulated by a body force field. Due to the presence of the blades, the flow-fields between any row blade passages can be different. Therefore, a three-dimensional flow-field in a blade row can be composed of an infinite number of axisymmetric flow-fields. This idea is illustrated in **Figure A.31**.

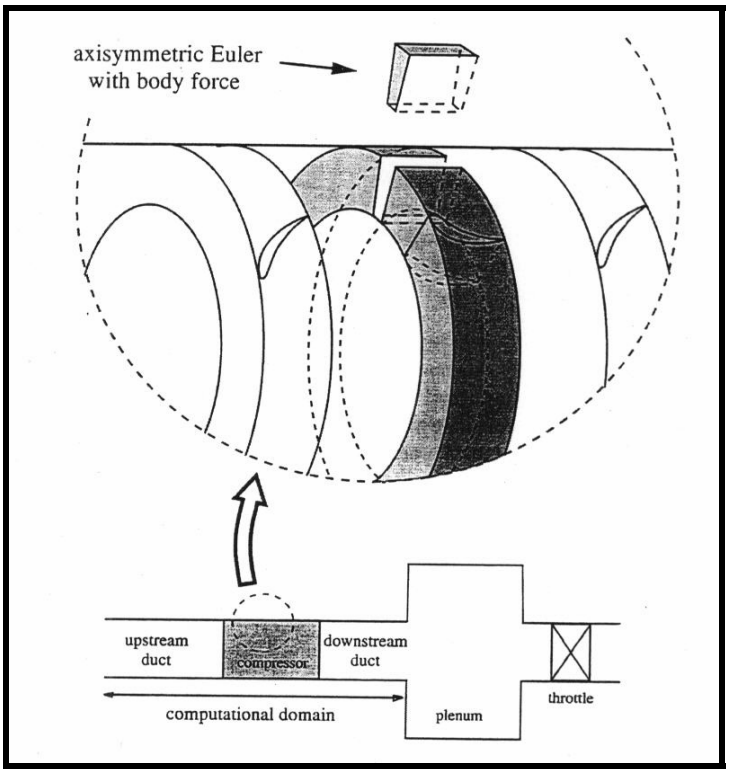


Figure A.31: Typical Blade Row Characteristics.

The requirements on the body force field are:

- In steady axisymmetric conditions, the body force should be capable of reproducing the required pressure rise and flow turning;
- The body force should be capable of responding to the flow disturbances for both the steady situations (e.g. inlet distortions) and unsteady flow situations;
- The body force field is formulated from the following given compression system characteristics:
    - Pressure rise characteristic in each blade row,

- Exit relative flow angle, and
- Blade metal angle distribution.

A notional characteristic set is illustrated in **Figure A.32**. This type of characteristic is similar to that used in the 1-D models such as DYNTECC (see ***Section A.1.3.2***).

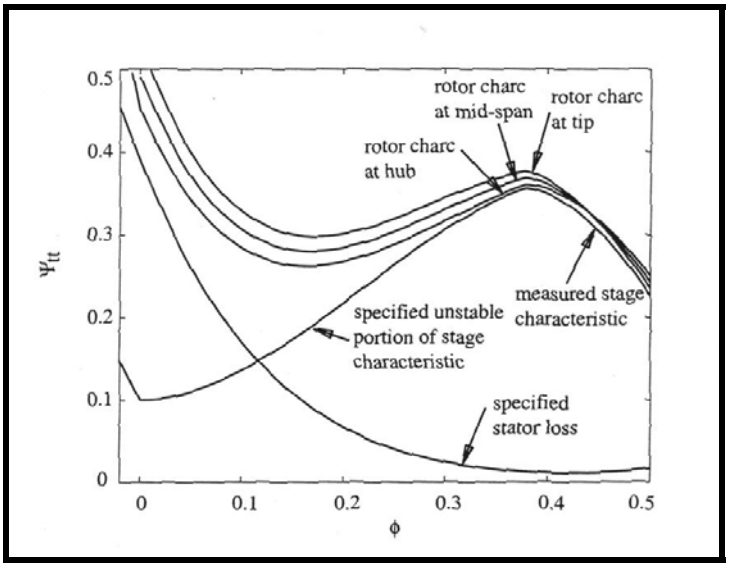


Figure A.32: Euler Approach with Body Forces.

Potential Benefits

Having a model to emulate the inception of rotating stall or surge provides a means for improving the operational characteristics of any compression system. Understanding the mechanisms of rotating stall inception will provide compression system designers a means to prevent or delay the onset of this type of instability. A strength of this type of modeling technique is that the model can be used to investigate the interactions between the compressor and other components. Some types of investigations that could be conducted are:

- Interaction between the inlet and the compressor with distortions and its impact on the performance and stability margin;
- Hot gas ingestion into the engine and its impact on stability;
- The behavior of an inlet vortex in an intake and its impact on the performance and stall margin; and
- The model could become a component in an engine system to model the dynamic behavior of a whole engine under various dynamic situations.

Cited Examples

- Gong, Y., "A Computational Model for Rotating Stall and Inlet Distortions in Multistage Compressors", GTL Report #230, March 1999, Gas Turbine Laboratory, Massachusetts Institute of Technology, Cambridge, MA. **[A.17]**
- Longley, J.P. et al., "Effects of Rotating Inlet Distortion on Multistage Compressor Stability", ASME Paper # 94-GT-220, June 1994. **[A.18]**

In the work by Gong, the model is used to simulate stall inception. Two major inception types have been experimentally identified: modal waves and spikes. Modal waves are exponentially growing long wavelength (length scale comparable to the annulus) small amplitude disturbances. Modal waves penetrate the whole compressor in the axial direction, so they can be detected by sensors at any locations at the inlet, exit, or with the compressor. The other inception mechanism is the growth of localized non-linear short wavelength (with length scale of several blade pitches) disturbances, often referred to as 'spikes'. The inception starts as one or several spike-shaped finite amplitude disturbances within the tip region of a particular stage. Usually, the disturbance develops into a large full span stall cell within three-to-five rotor revolutions. The simulation was executed such that a short wavelength stall inception was initiated by a spike-shaped disturbance.

The flow coefficient traces shown in **Figure A.33**, as taken from the tip region of the first rotor inlet of a four-stage GE low-speed compressor, show that the disturbance is sustained and the disturbance leads to compressor stall subsequently. The overall inception is similar to the measurements as illustrated in **Figure A.34**. A quantitative comparison shows that the stall inception initiated by the spike-shaped disturbance has an initial disturbance rotating speed of 83% of rotor speed, and a transition time of about three-rotor revolutions and compares reasonably well with experimental results.

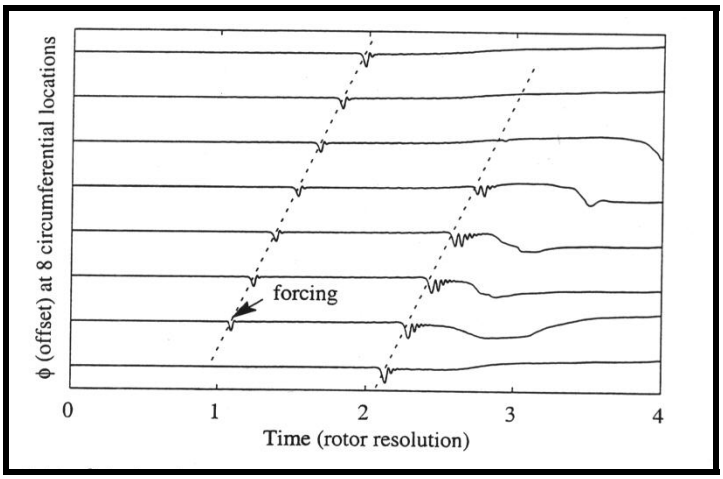


Figure A.33: Model Prediction of Stall Inception Process for Short Wavelength (Spike) Stalls.

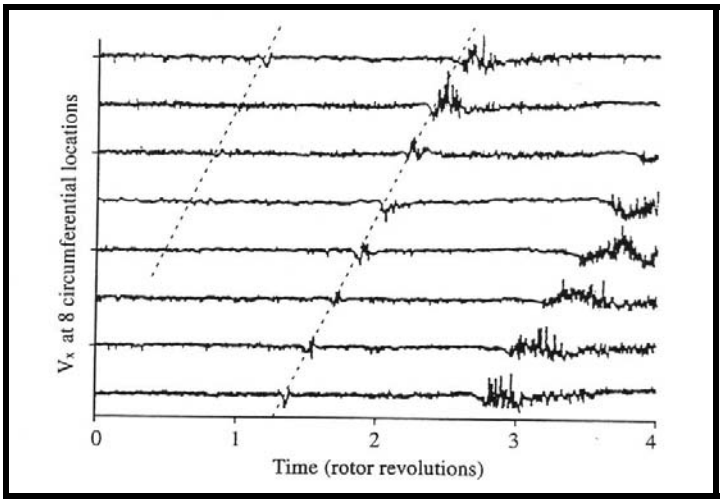


Figure A.34: Experimental Data of Stall Inception Process for Short Wavelength (Spike) Stalls.

In multi-spool engines, rotating stall in an upstream compressor will impose a rotating distortion on the downstream compressor, thereby affecting its stability margin. Inlet distortion can modify the flow rate at which rotating stall occurs, degrading the range over which stable operation is possible. The investigation reported by Longley [A.18] addresses the relationship between speed and direction at which the inlet distortion pattern rotates around the compressor annulus and the severity of the degradation in stability margin. Imposition of a rotating distortion can also be thought of as a type of forced response experiment to probe the compressor dynamic behavior. The measured mean flow coefficients at stall inception as a function of screen rotation speed, the means by which rotating distortion was imposed, for the GE four-stage compressor is presented in **Figure A.35**. The calculations from the modeling technique, described in the paper by Gong are shown in the figure as solid lines. Both the data and the model indicated that there is less stability margin available with co-rotating distortion than with the counter-rotating screen.

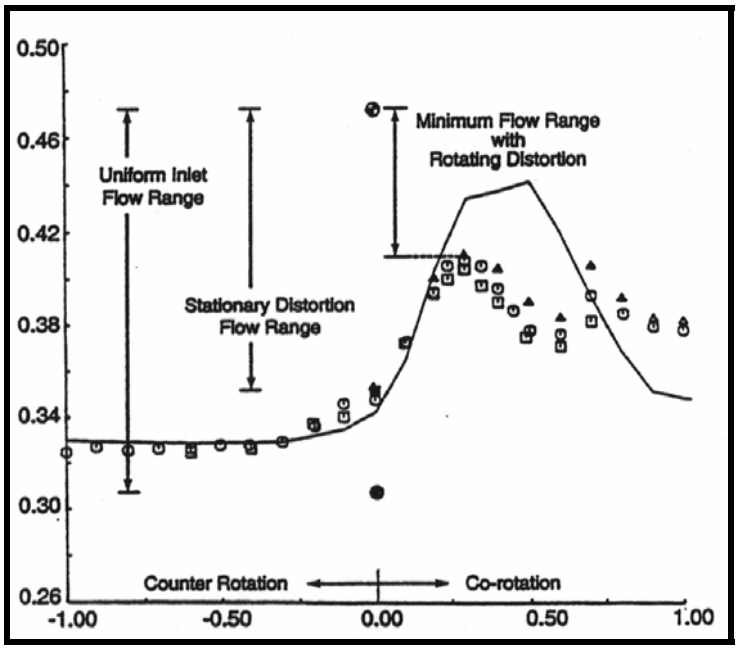


Figure A.35: The Effects of Co- and Counter-Screen Rotation on Compressor Stability Limit.

Limitations of Chosen Modeling Technique

This modeling technique has its strengths and weaknesses. The approach allows for a relatively quick computation of distortion as compared to traditional CFD turbomachinery, but because there is a level of empiricism the flow-field may not be accurately represented at a detailed level suitable for design. This modeling technique is best suited for studying potential design changes that can be quantified as to their effects on blade performance and system response to inlet distortion effects.

A.3.2.3 Multi-Disciplinary Interaction for Durability

Traditionally, aeropropulsive and structural performance have been designed separately and later mated together via flight-testing. In today's atmosphere of declining resources, it is imperative that more productive ways of designing, optimizing and verifying aeropropulsive performance and structural aerodynamic interaction are made available to the aerospace industry. One method of obtaining a more productive design and evaluation capability is through numerical simulations. Recent advances in turbomachinery design are leading to very high thrust, lightweight engines that challenge all fronts of technology development. High temperature super alloys with single crystal construction offer tremendous resilience in extremely harsh turbine engine operating environments. Similarly, the high-bypass wide-

chord, lightweight hollow, or composite, fan blade offers tremendous strength during bird strikes, hale ingestion, and surge cycles.

Modeling Techniques Used

There are two technical approaches for analyzing aerodynamic-structural interactions. The more traditional approach is to use a computational fluid dynamic code to obtain aerodynamic forcing functions, and then pass that information to a finite-element structural code. The second approach is to use a code that integrates both the structures and aerodynamics. This approach has been demonstrated with a code known as ALE3D, developed by Lawrence Livermore National Laboratory. This code is capable of characterizing fluid and structural interaction for components such as the combustor, fan and stators, inlet and nozzles. This code solves the 3-D Euler equations and has been applied to several aeropropulsive applications, such as a supersonic inlet and a combustor rupture simulation.

ALE3D was developed from a version of DYNA3D. It uses the basic Lagrangian finite element techniques developed there but has not maintained an identical set of algorithms as the two code efforts evolved along different paths. The fluid dynamic treatment of solid elements has been completely rewritten. However, the coding and the available models for treating beam and shell elements have been kept consistent with the equivalent DYNA3D models, although only a subset are currently available. Fluid mechanics and ALE techniques from JOY and CALE were modified for application to unstructured meshes and incorporated into ALE3D. Thermal and structural analysis techniques are generally developed first in DYNA3D and TOPAZ3D then migrated to ALE3D as required. (See **[A.19]** for specific references to the Lawrence Livermore suite of codes listed in this paragraph).

The basic computational step consists of a Lagrangian step followed by an advection, or remap step. This combination of operations is formally equivalent to an Eulerian solution while providing increased flexibility and, in some cases, greater accuracy. In the Lagrangian phase, nodal forces are accumulated and an updated nodal acceleration is computed. Following DYNA3D, the stress gradients and strain rates are evaluated by a lowest order finite-element method. At the end of the Lagrangian phase of the cycle, the velocities and nodal positions are updated. At this point, several options are available. If the user wishes to run the code in a pure-Lagrangian mode, no further action is taken and the code proceeds to the next time step. If a pure-Eulerian calculation is desired, the nodes are placed back in their original positions. This nodal motion or relaxation generates inter-element fluxes that must be used to update velocities, masses, energies, stresses and other constitutive properties. This re-mapping process is referred to as advection. Second-order schemes are required to perform this operation with sufficient accuracy. In addition, it is not generally adequate to allow advection only within material boundaries. ALE3D has the ability to treat multi-material elements, thus allowing relaxation to take place across material boundaries.

Potential Benefits

With this coupled approach, a numerical tool is available to the aeropropulsion community that will allow fully interactive analysis between the traditionally uncoupled aerodynamics and structural disciplines. A system level approach, which would allow internal engine component and sub-components (compressor stage or blade row) to interact with the full turbine engine system and thus with external aircraft structures, is envisioned. If analysis of an internal engine component were desired, a more traditional CFD approach would be available through zooming.

Cited Example

- Nazir, J., Couch, R. and Davis, M., “An Approach for the Development of an Aerodynamic-Structural Interaction Numerical Simulation for Aeropropulsion Systems”, ASME Paper # 96-GT-480, June 1996. **[A.19]**

An engine-inlet configuration associated with the HSCT program was analyzed in terms of inlet unstart and the effect of the regurgitated shock wave. Inlet start is a complex three-dimensional phenomenon where a supersonic flow, which comes through the inlet, is stabilized to some acceptable subsonic condition before entering the fan. This produces a shock wave that sits strategically somewhere in the HSCT inlet, thus creating a transition zone from supersonic flow to subsonic flow within the inlet itself. If, for whatever reason, the engine undergoes a transient such as an engine surge, the stable shock wave will be disrupted, and may become unstable, possibly spilling out around the engine. This bubble or plume will spread and produce loading on the surrounding structure, such as the wing, and can affect the aircraft attitude-control surfaces. These will try to compensate for such pulse loading, as illustrated in **Figure A.36**. If large enough, the bubble or plume could be sucked in by the adjacent engine, causing it also to unstart. This will obviously intensify the dynamics for the control system to compensate, thus requiring a thorough understanding of this phenomenon.

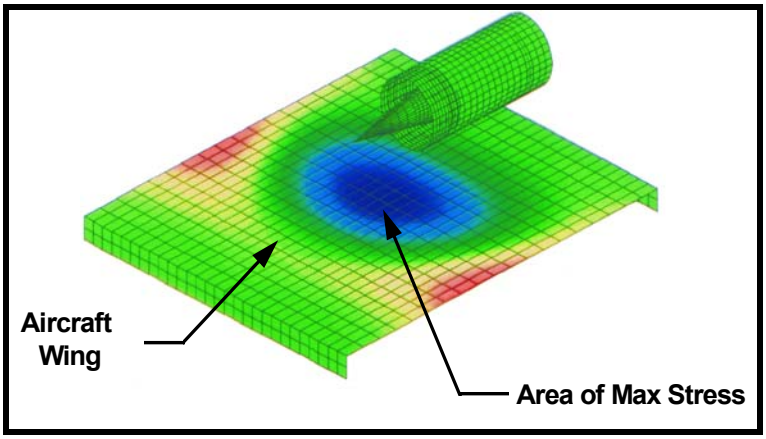


Figure A.36: Effect of Inlet Unstart on Aircraft Wing Assembly.

The initial approach has been to analyze this coupled aerodynamic-structural interaction with a de-coupled numerical technique. The approach that was taken was to model the surge cycle frequencies and intensity using a one-dimensional compression system model, DYNTECC [A.2], for a typical high-pressure compressor. This scaled pressure loading at the fan face was then introduced as a boundary condition to ALE3D, which characterized the inlet steady state shock location in a three-dimensional inlet. The appropriate boundary conditions were applied and the equilibrium flow was obtained as described in the previous section. The surge conditions were then applied as a time-variant one-dimensional pressure boundary condition. The initial spike and rapid drop-off occurs within the first 10 to 15 milliseconds after the event is initiated. The highly cyclic nature of the blowdown part of the cycle was not represented by the boundary condition. However, the cyclic oscillations cease to play a role in the inlet unstart once it has begun.

Limitations of Chosen Modeling Technique

The major limitation with this technique was the need to loosely couple the compression system code to the ALE3D simulation. The output of the DYNTECC code had to be used a boundary condition to the ALE code. Full interaction was not realized with this technique. In addition, when using a highly coupled approach, some compromises had to be made in the development of the code. These compromises can manifest themselves in reduced capability that might otherwise be available in a separate computational fluid dynamics code or finite-element structural code.

A.3.2.4 Component Aerodynamic Design

Turbomachinery aerodynamicists have long realized that the flow within multi-stage turbomachinery is complex. In addition, to the non-deterministic, small-scale chaotic unsteadiness due to turbulence, at a large scale the flow is also unsteady and periodic from blade passage to blade passage. The flow features associated with these large scales are deterministic. It is because of the unsteady deterministic flow that turbomachinery is able to either impart or extract energy from a flow. The challenge is to develop an analytical model of sufficient fidelity to address key design issues. At the same time, the cost and time of executing a simulation based on the model, and the cost of acquiring and maintaining the empirical database that underpins the model must be compatible with the design environment.

Modeling Techniques Used

Within the last ten years, there has been a steady infusion of 3-D CFD-based models into axial flow multi-stage turbo-machinery design systems. In part this has been due to the useful role these models have played in the analysis of isolated blade rows. The performance level of the fan rotor of the current generation of high-bypass-ratio turbofan engines is directly tied to the advances made in the use of 3-D CFD-based models. A through-flow or a quasi-three-dimensional system may provide the initial fan-rotor geometry, but the final shape of the rotor is evolved exclusively using 3-D CFD-based models. The rotor shape is tailored to control the interaction of shock waves with the blade surface boundary layers to avoid separation and to minimize loss. In addition, using these 3-D CFD models one is able to resolve the tip clearance flow as well as the stator hub leakage flow, and establish their impact on aerodynamic performance. From this activity guidelines have been established for use in aerodynamic design.

The 3-D CFD-based models that were initially introduced into multi-stage axial flow turbo-machinery design systems ignored the impact of the unsteady, deterministic flow existing within axial flow multi-stage turbo-machines. The unsteady, deterministic flow is produced by the surrounding blade rows and its frequency content is linked to shaft rotational speed. These 3-D models have two forms. The first is simply an isolated 3-D blade row CFD-based model linked to a through-flow code. The initial geometry and the inlet and exit flow conditions to the blade row being designed are established by the through-flow or quasi-3-D system. The established inlet and exit flow conditions to a blade row set the boundary conditions for the 3-D model. 3-D simulations are executed with geometry updates until a blade configuration is found whose inlet and exit flows closely match that of the through-flow system. The aerodynamic matching of stages within a machine is established by the through-flow system.

Another approach is based on the flow model derived from the work of Adamczyk [A.11]. This model and its off-shoots have proven useful in design applications. The model developed by Adamczyk is referred to as the average-passage flow model. This flow model describes the time-averaged flow-field within a typical passage of a blade row embedded within a multi-stage configuration. The resulting flow-field is periodic over the pitch of the blade row of interest. The average-passage flow model is an analysis model, as are all the others that have been referred to thus far. This means that geometry is the input and the output is the flow-field generated by the geometry.

When using the average-passage flow model in aerodynamic design, the initial geometry is defined by a through-flow system. During the design process, geometry updates are done exclusively based on simulation results from the average-passage model. No output from an axisymmetric through-flow system or a data match that implicitly or explicitly sets the aerodynamic matching of stages within a turbomachine is provided to the average-passage simulations. The credibility of an average-passage flow simulation is not tied to aerodynamic matching information provided by a through-flow system or a data match. The credibility is tied to the models used to account for the effects of the unsteady flow environment on the average-passage flow-field. The effect of the unsteady deterministic flow-field on aerodynamic matching of stages is accounted for by velocity correlation within the momentum equations associated with the

average-passage flow-field and by a velocity total enthalpy correlation within the energy equation associated with the average-passage flow-field. The term unsteady deterministic refers to all time-dependent behavior that is linked to shaft rotational speed. All unsteady behavior not linked to shaft rotational speed is referred to as non-deterministic.

Potential Benefits

Up until recently (the late 90's), the aerodynamic design of most axial flow multi-stage turbomachinery has been executed using various axisymmetric flow models. The use of these models is iterative. A design has been executed based on the existing database. Data obtained from tests of the fabricated hardware are used to update empirical correlations embedded within these models. The updated axisymmetric flow model is then used to support the next design of the configuration. This bootstrap approach to the aerodynamic design of axial flow multi-stage turbomachinery results in some truly impressive machines, as evidenced by the aerodynamic performance they achieve. However, because of strong economic forces, the turbomachinery industry has been forced to re-examine this iterative approach to aerodynamic design. These economic forces mandated that industry reduce the time and cost of developing a turbomachinery component and, at the same time, required the design of machinery whose performance goals lay outside the then-existing experience base. As a result, a strong need to develop a new methodology, for executing the aerodynamic design of axial flow multi-stage turbomachinery, has arisen. The foundation of this new methodology requires aerodynamic models whose resolution is greater than that of the axisymmetric flow models. These models have to allow for overnight turnaround using computer resources compatible with the design environment. Finally, the model has to be capable of addressing the aerodynamic issues associated with the design of advanced configurations.

Cited Example

- Adamczyk, J.J., "Aerodynamic Analysis of Multistage Turbomachinery Flows in Support of Aerodynamic Design", 1999 International Gas Turbine Institute Scholar Lecture, Transactions of the ASME, Journal of Turbomachinery, Vol. 122, April 2000, pp. 189-217. **[A.11]**

Figure A.37 shows the measured pressure rise characteristic along with simulation results at a flow coefficient of 0.395 and 0.375 for the Lewis low-speed axial compressor (LSAC). LSAC is a four-stage machine with an inlet IGV, which is representative of the rear stages of a high-pressure (HP) compressor. The compressor is of a modern design employing hub-shrouded stators with end-bends. The four stages are geometrically identical. The simulation accounted for the rotor tip clearance. The simulation did not include the stator hub cavities nor did the simulations account for stator hub leakage.

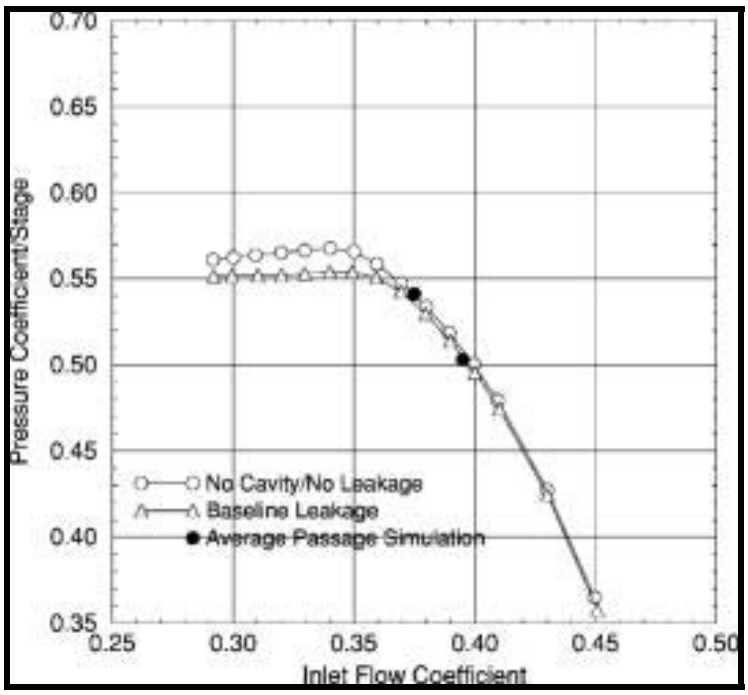


Figure A.37: Overall Performance Characteristic.

At the flow coefficient of 0.395 (which is near the measured peak efficiency operating point) and at the flow coefficient of 0.375, the simulation results are in good agreement with the measurement. An attempt to simulate an operating condition near peak pressure failed to converge. The simulation did not account for casing treatment over the first rotor present in the experiment. Tests with the casing treatment removed show that the compressor stalls at a flow coefficient near the peak pressure point of the characteristic. **Figure A.38** shows the measured static pressure rise characteristic for each stage along with results from the simulations. The agreement between the simulation results and the data is very good. For the flow coefficient of 0.395, **Figure A.39** shows plots of the total and static pressure coefficient, the axial and absolute tangential velocity, and the absolute and relative flow angle as a function of span for the simulation and the experiment. The plots are for an axial location behind the second stator. Once again, the agreement between the simulation results and the data is good. The slight difference between the static pressure coefficient derived from the simulation and that measured inboard at 40 percent span is unknown.

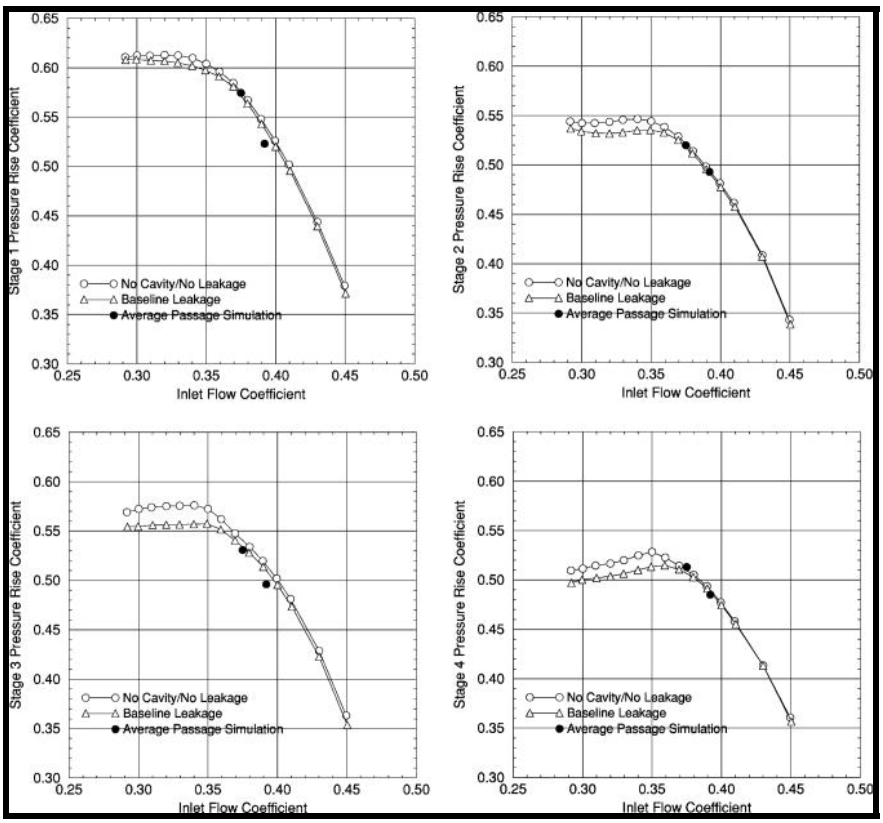


Figure A.38: Individual Stage Pressure Rise Coefficient.

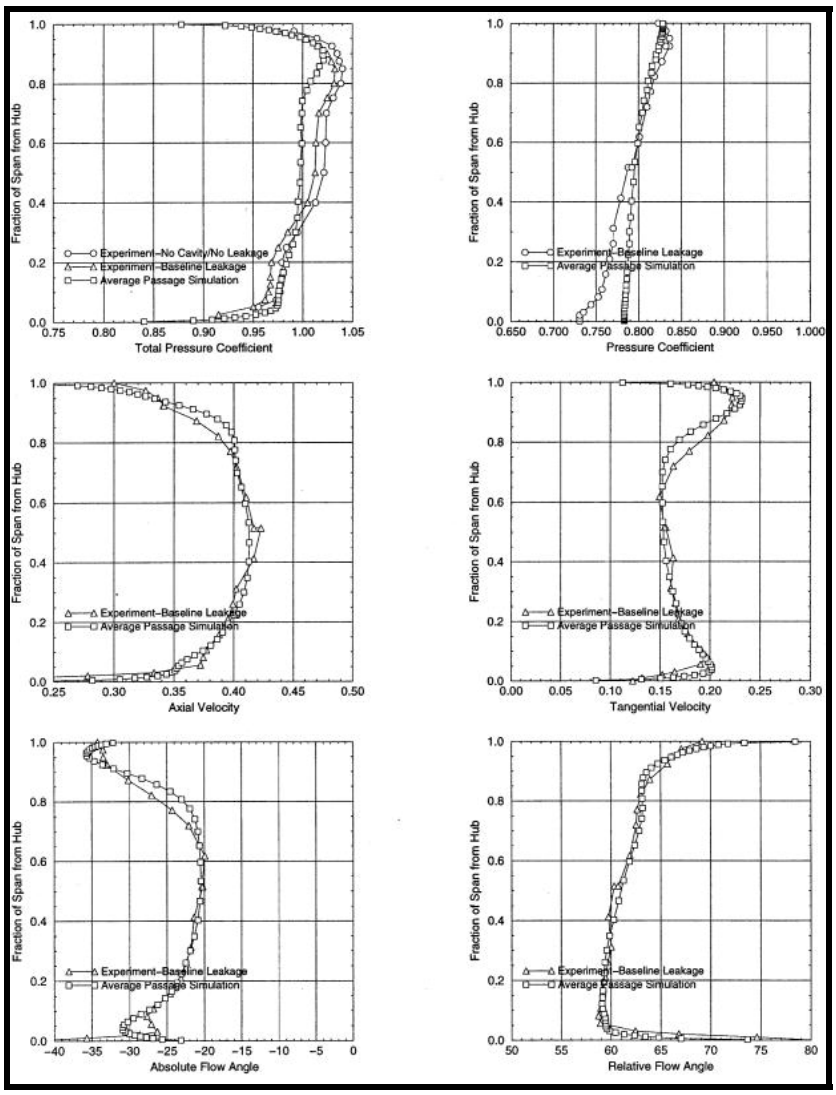


Figure A.39: Axisymmetric Flow Variables, Exiting the Second Stage Rotor.

Limitations of Chosen Modeling Technique

There is no doubt that 3-D CFD-based models with aerodynamic input from through-flow models have provided credible designs. These models will continue to provide credible designs as long as the design parameters are within the bounds of the database underpinning well-calibrated through-flow or quasi 3-D design systems. However, reliance on through-flow or quasi-3-D models to set key aerodynamic design parameters such as the aerodynamic matching of stages greatly impedes the utility of these models. Specifically, they are limited in their ability to uncover the fluid mechanics controlling the performance of multi-stage axial flow turbomachines at design and off-design operating conditions. The objective of 3-D CFD-based modeling should be to break free of the dependence on through-flow or quasi 3-D design systems except for providing the initial geometry. If such 3-D CFD-based models are proven credible, their use would allow aerodynamic designs of machinery whose parameters lay outside the bounds of the database, allowing through-flow systems to be used with confidence. This would include both design and off-design operation, and in the case of compressors, prediction of maps including the surge line.

A.4 REFERENCES

[A.1] Rodriguez, C.G. and O'Brien, W.F., "Unsteady, Finite-Rate Model for Application in the Design of Complete Gas-Turbine Combustor Configurations", Design Principles and Methods for Aircraft Gas Turbine Engines, RTO-MP-8, February 1999.

[A.2] Hale, A.A. and Davis, Jr., M.W., "Dynamic Turbine Engine Compressor Code, DYNTECC – Theory and Capabilities", AIAA Paper # 92-3190, June 1992.

[A.3] Davis, Jr., M.W. et al., "Euler Modeling Techniques for the Investigation of Unsteady Dynamic Compression System Behavior", Loss Mechanisms and Unsteady Flows in Turbomachines, AGARD-CP-571, January 1996.

[A.4] Garrard, G.D., "ATEC: The Aerodynamic Turbine Engine Code for the Analysis of Transient and Dynamic Gas Turbine Engine System Operations – Part 1: Model Development", ASME Paper # 96-GT-193, June 1996.

[A.5] Garrard, G.D., "ATEC: The Aerodynamic Turbine Engine Code for the Analysis of Transient and Dynamic Gas Turbine Engine System Operations – Part 2: Numerical Simulations", ASME Paper # 96-GT-194, June 1996.

[A.6] Lefebvre, A.H., "Fuel Effects on Gas Turbine Combustion – Ignition, Stability, and Combustion Efficiency," Journal of Engineering for Gas Turbines and Power, Vol. 107, January 1985, pp. 24-37.

[A.7] Derr, W.S. and Mellor, A.M., "Recent Developments," in *Design of Modern Turbine Combustors*, edited by A.M. Mellor, Academic Press, Harcourt Brace Jovanovich, New York, NY, © 1990.

[A.8] Shahrohki, K.A. and Davis, Jr., M.W., "Application of a Modified Dynamic Compression System Model to a Low-Aspect Ratio Fan: Effects of Distortion", AIAA-95-0301, Presented at the 33rd Aerospace Science Meeting, Reno, NV, January 1995.

[A.9] Hale, A.A. and O'Brien, W.F., "A Three-Dimensional Turbine Engine Analysis Compressor Code (TEACC) for Steady-State Inlet Distortion," *Journal of Turbomachinery*, Vol. 120, July 1998, pp. 422-430.

[A.10] Stewart, M.E.M., "Axisymmetric Aerodynamic Numerical Analysis of a Jet Engine", ASME Paper # 95-GT-338, June 1995.

[A.11] Adamczyk, J.J., "Aerodynamic Analysis of Multistage Turbomachinery Flows in Support of Aerodynamic Design", 1999 International Gas Turbine Institute Scholar Lecture, Transactions of the ASME, Journal of Turbomachinery, Vol. 122, April 2000, pp. 189-217.

[A.12] Suresh, A. et al., "Analysis of Compressor-Inlet Acoustic Interactions Using Coupled Component Codes", AIAA Paper # 99-0749, 1999.

[A.13] Costura, D.M. et al., "A Model for Combustor Dynamics for Inclusion in a Dynamic Gas Turbine Simulation Code", AIAA Paper # 97-3336, Presented at the 33rd AIAA/ASME/SAE/ASEE Joint Propulsion Conference, July 6-9, 1997, Seattle, WA.

[A.14] Garrard, G.D., Davis, Jr., M.W., Wehofer, S. and Cole, G., "A One-Dimensional Time Dependent Inlet/Engine Numerical Simulation for Aircraft Propulsion Systems", ASME Paper # 97-GT-333, June 1997.

[A.15] Abdel-Fattah, A.M., "Response of a Turbofan Engine Compression System to Disturbed Inlet Conditions", Journal of Turbomachinery, Vol. 119, No. 4, October 1997.

[A.16] Hale, A.A., Chalk, J., Klepper, J. and Kneile, K., "Turbine Engine Analysis Compressor Code: TEACC – Part II: Multi-Stage Compressors and Inlet Distortion", AIAA Paper # 99-3214, Presented at the 17th AIAA Applied Aerodynamics Conference, June 1999.

[A.17] Gong, Y., "A Computational Model for Rotating Stall and Inlet Distortions in Multistage Compressors", GTL Report #230, March 1999, Gas Turbine Laboratory, Massachusetts Institute of Technology, Cambridge, MA.

[A.18] Longley, J.P. et al., "Effects of Rotating Inlet Distortion on Multistage Compressor Stability", ASME Paper # 94-GT-220, June 1994.

[A.19] Nazir, J., Couch, R. and Davis, M., "An Approach for the Development of an Aerodynamic-Structural Interaction Numerical Simulation for Aeropropulsion Systems", ASME Paper # 96-GT-480, June 1996.