A COMPARATIVE STUDY OF HIGH AND LOW FIDELITY FAN MODELS FOR TURBOFAN ENGINE SYSTEM SIMULATION

JOHN A. REED

The University of Toledo
Toledo, OH 43606, USA

ABDOLLAH A. AFJEH

The University of Toledo
Toledo, OH 43606, USA

ABSTRACT

In this paper, a heterogeneous propulsion system simulation method is presented. The method is based on the formulation of a cycle model of a gas turbine engine. The model includes the nonlinear characteristics of the engine components via use of empirical data. The potential to simulate the entire engine operation on a computer without the aid of data is demonstrated by numerically generating "performance maps" for a fan component using two flow models of varying fidelity. The suitability of the fan models were evaluated by comparing the computed performance with experimental data. A discussion of the potential benefits and/or difficulties in connecting simulations solutions of differing fidelity is given.

Key Words: Numerical, Simulation, Turbofan, Modeling, Engine, Fidelity.

INTRODUCTION

Numerical simulation methods for an aircraft propulsion system generally break down the system into subsystems, or components, and analyze through conservative physical laws the change in state properties (pressure, temperature, mass flow, etc.) at the entrances and exits of each of the engine components (fan, compressor, etc.). The interplay among various engine components is usually modeled by requiring continuous values for flow properties and fluxes at the interfaces of successive components. This procedure results in a set of coupled differential equations which can then be solved for the state variables in a propulsion system. The empirical data describing the operating characteristics of each of the engine components within their operating range serves as closure to these equations.

One significant drawback to this approach is the requirement that empirical data be available for each of the components in the simulation. This requirement can pose significant limitations in the simulation, especially during the early stages of new component design when no experimental data exist and it is important to determine the relevant factors to the design. Furthermore, the available data are specific to a given design and cannot be reliably extended from one engine to another. Currently, new design data are generated by scaling existing data. But this is not always accurate and does not allow for dynamic interaction among system components. Thus, the quality of a design can only be verified by building the components and testing them in operation. Typically, several iterations of this design-test-redesign cycle are needed to achieve the desired performance. This can make component design a very costly and time consuming process.

The challenge then, is to develop a numerical simulation scheme which would allow engine designers to generate "empirical" data from the existing geometry data [1]. To capture the physics of the flow, one often needs to produce detailed flow characteristics within a component and then transform this into a form that can be used by the low fidelity system simulation model. When this feature is incorporated into an engine simulation, the user will have the option to select the level of fidelity for the different processes in the simulation that provides the needed information most efficiently. This concept of moving from one level of fidelity to another within a simulation is known as *zooming*, so-called because it allows the simulation user to change the resolution of the simulation in much the same manner as zooming in a graphical computer program changes the resolution of target objects of computer screen.

Zooming can be quite useful in at least two situations. The first case occurs when an engine simulation requires information for a component at a point of operation which lies off an available performance map, or out-of-range of the measured data. By invoking a high fidelity solver to generate performance data at this point and then transforming the data into a form compatible with the engine model, the simulation may be continued.

component was selected as a test case. The fan performance was computed using two flow solvers, the 3D flow solver, ADPAC [4], and an axisymmetric compressor code, [5]. For each of the remaining E^3 components performance maps were constructed from the available experimental data. This step was necessary in order to provide the engine system simulation with accurate data. In this way, the entire engine simulation results could be compared with the experimental results.

FAN COMPONENT MODELS

Since the 2D code could not model the complex fan geometry directly, a model was developed which treats the fan as two fans of different sizes. For the purposes of computaions, the flow entering the fan was divided (conceptually) into "core" and "by-pass" flows. The core flow is identified as that flow contained between the streamline coincident with the axis of the engine and the "splitting" streamline (stream surface), the streamline that splits at the stagnation point on the leading edge of the part of engine that physically separates the engine core flow from that of the by-pass flow. The location of the splitting streamline was determined from an analysis of the actual fan flow at the design operating point and held fixed for all other operating conditions. The by-pass fan flow is defined as the flow passing through an annular tube defined by the splitting stream surface and the cowl inner surface. No interactions between the core and by-pass fan flows were permitted. In adddition, the effect of downstream components on the fan flow was neglected for both the core and by-pass fan models.

The fan performance was also computed using the 3D flow solver, ADPAC [4]. In the experimental data taken for the

E^3 fan, there were no downstream disturbances from the engine components, this was accounted for by modifying the computational grid representing the flow. The fact that the flow field model did not have any disturbances downstream was used to determine the correct downstream boundary condition for the flow solver. Without any flow disturbances, the total conditions for the flow were considered to be the same from the fan exit to the downstream boundary in the ADPAC model. Ordinarily, however, the downstream boundary conditions would be expected to be determined from the simulation of the core and by-pass duct. This capability is not currently available. Thus, for the purposes of this work the experimental data at the fan exit plane were duplicated in the definition of the downstream boundary conditions.

RESULTS

The state variables of interest at the inlet and exit of the fan are the total pressures, the total temperature, fan efficiency, and the mass-flow rate. The inlet conditions are specified by the boundary conditions for the engine, and what remains is to calculate the exit conditions. Assuming no bleed, the mass-flow rate remains constant and is used along with the low-speed spool rotational speed to extract the total pressure ratio, and the thermodynamic efficiency across the fan from the fan performance map. The compted fan performance using the 2D fan flow model is shown in Figs. 3 and 4 along with the corresponding experimental data. As it can be seen the agreement is quite good for both the core and by-pass flows. Similar agreement with experimental data was observed for 3D simulations. However, since the computational costs of 3D simulations were prohibitive (about 20 hours of CPU time on a Cray YMP supercomputer per operating point) only significant

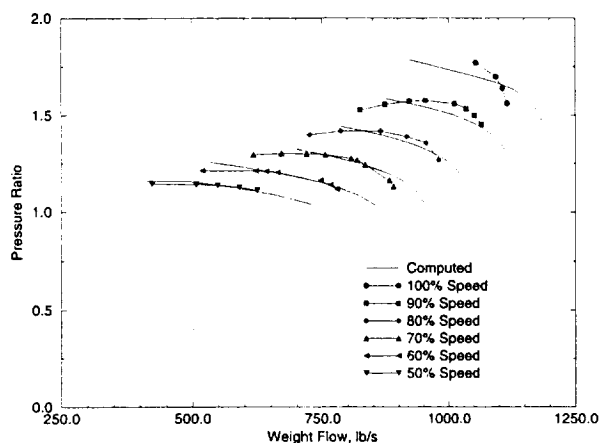FIGURE 3. CORE FLOW FAN PERFORMANCE
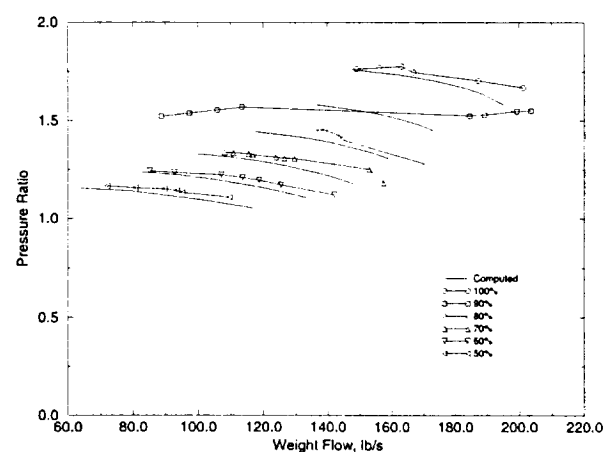

FIGURE 4. BYPASS FLOW FAN PERFORMANCE

Table 1 - Comparison of fan performance experimental data and simulation results

Data Normalized to 100% speed	Experimental		Simulation-2D		Simulation-3D	
	Efficiency	Pressure ratio	Efficiency	Pressure ratio	Efficiency	Pressure ratio
0.936	0.962	1.056	0.965	1.070	0.970	1.074
0.963	0.988	1.054	0.987	1.055	0.982	1.012
0.975	1.005	1.042	1.005	1.040	1.005	1.006
0.988	1.007	1.025	1.019	1.030	1.006	1.029

operating conditions were computed, that is, only operating points close to the design point were computed. Table 1 contains selected computed data using ADPAC solver. For comparison purposes, the corresponding experimental and 2D simulation results are also included in the Table.

Using the numerically generated fan maps, a balanced engine operating at the design point was achieved, as expected. The engine performance results compared very well with the experimental E^3 simulation.

Discrepancies between the experimental and ADPAC data can possibly be attributed to the fact that the E^3 geometry used in the ADPAC model was modified from the actual fan geometry in the experiment. Also, the fact that no fan stator geometry was modeled in the present simulation may have contaminated the results.

CONCLUSIONS

A heterogeneous propulsion system simulation model consisting of differing levels of fidelity was developed and validated using experimental data. The feasibility of utilizing such a system has been established by connecting 2D and 3D flow solvers with a "zero-dimensional" cycle engine system simulation in the analysis of E^3 turbofan engine. It has been shown that the addition of different levels of fidelity can offer potentially great advantages in engine simulation and engine design. But, with those advantages come significant difficulties and increasing complexities to the simulation. It has been shown that flow models with different fidelity may produce practically the same results and thus complex 3D computations may not be necessary to generate engine data. Simpler, but proven, models may require much less computational effort and should be preferred in an optimum simulation strategy. level There still remains much work to do in determining

the optimum method for effectively and accurately exchanging data between the different simulation levels

ACKNOWLEDGMENTS

REFERENCES

[1] R. W. Claus, A. L. Evans, J. K. Lytle and L. D. Nichols, "Numerical Propulsion System Simulation," *Computing Systems in Engineering* 2, 357-364 (1991).

[2] D. Y. Davis and E. M. Stearns, "Energy efficient engine - flight propulsion system final design and analysis," NASA CR-168219, contract report prepared by General Electric Company, August 1985.

[3] J. A. Reed and A. A. Afjeh, "Development of an Interactive Graphical Propulsion System Simulator," AIAA paper 94-3216. 30th AIAA/ASME/SAE/ASEE Joint Propulsion Conference, Indianapolis IN, June 1994.

[4] E. J. Hall, R. A. Delaney and J. L. Bettner, " Investigation of advanced Counterrotation blade configuration concepts for high speed turboprop systems, task II - unsteady ducted propfan analysis (computer program user's manual)," NASA CR-187105, 1991.

[5] J.M. Crouse "Off-Design Axisymmetric Flow Through Axial Compressors," NASA TM 812323, 1980.