Chinese Journal of Aeronautics, (2017), xxx(xx): xxx–xxx

Chinese Society of Aeronautics and Astronautics
& Beihang University
Chinese Journal of Aeronautics
cja@buaa.edu.cn
www.sciencedirect.com

Development of a coupled supersonic inlet-fan Navier–Stokes simulation method

Qiushi LI [a,b,c], **Yongzhao LYU** [a,b], **Tianyu PAN** [a,b,c,*], **Da LI** [a,b], **Ha'nan LU** [a,b], **Yifang GONG** [d]

[a] *National Key Laboratory of Science and Technology on Aero-Engine Aero-Thermodynamics, Beihang University, Beijing 100191, China*
[b] *School of Energy and Power Engineering, Beihang University, Beijing 100191, China*
[c] *Collaborative Innovation Center of Advanced Aero-Engine, Beihang University, Beijing 100191, China*
[d] *GL-Turbo Compressor Company, Wuxi 214106, China*

Received 8 July 2016; revised 29 November 2017; accepted 29 November 2017

KEYWORDS

Body force model;
Coupled simulation;
Rapid numerical method;
Supersonic inlet-fan;
Viscous flow

Abstract A coupled supersonic inlet-fan Navier–Stokes simulation method was developed by using COMSOL-CFD code. The flow turning, pressure rise and loss effects across blade rows of the fan and the inlet-fan interactions were taken into account as source terms of the governing equations without a blade geometry by a body force model. In this model, viscous effects in blade passages can also be calculated directly, which include the exchange of momentum between fluids and detailed viscous flow close to walls. NASA Rotor 37 compressor test rig was used to validate the ability of the body force model to estimate the real performance of blade rows. Calculated pressure ratio characteristics and the distribution of the total pressure, total temperature, and swirl angle in the span direction agreed well with experimental and numerical data. It is shown that the body force model is a promising approach for predicting the flow field of the turbomachinery. Then, coupled axisymmetric mixed compression supersonic inlet-fan simulations were conducted at Mach number 2.8 operating conditions. The analysis includes coupled steady-state performance, and effects of the fan on the inlet. The results indicate that the coupled simulation method is capable of simulating behavior of the supersonic inlet-fan system.

1. Introduction

Over the last decades, Computational Fluid Dynamics (CFD) has been well developed, which can accurately simulate the flow field of the turbomachinery and revolutionize the aerodynamic design process of propulsion system components. Full annulus multi-row unsteady calculations through the

* Corresponding author at: National Key Laboratory of Science and Technology on Aero-Engine Aero-Thermodynamics, Beihang University, Beijing 100191, China.
E-mail address: pantianyu@buaa.edu.cn (T. PAN).
Peer review under responsibility of Editorial Committee of CJA.

ELSEVIER Production and hosting by Elsevier

https://doi.org/10.1016/j.cja.2017.11.011
1000-9361

turbomachinery subject to non-axisymmetric flow can be calculated directly to resolve effects of separations close to the walls on the compressor characteristic, which in some cases is the center of concern. For instance, the Hybrid Wing-Body (HWB) aircraft is extremely concerned lately, which is an alternative concept for the conventional tube-and-wing aircraft.[1] In the design process, it involves coupled calculations of the inlet and full-annulus fan blades, and it is important to include the effect of fan blades on the inlet and nozzle design. A direct coupling of the inlet and full-annulus fan blades in the computational domain is more realistic and accurate for the inlet-fan interaction, but the demand for the computer resources is prohibitively large, including the memory and the CPU time, and more than tens of flow simulations are usually required.[2] To save the computer resources, some CFD studies were conducted using conventional uniform-back-pressure boundary condition at the fan face to simulate flow for propulsion-airframe integration problems.[3–5] However, for integration problems with highly distorted flows at the fan faces, the assumption of uniform static pressure may not be valid. By implementing an actuator disk with a Navier–Stokes code, it is possible to simulate the flow field through fan blades without the actual blade geometry.[6] However, based on the two-dimensional assumption, the actuator disk model does not include the effect of swirl and requires the total pressure and total temperature change across the fan as input terms. Recently, the passage averaged body force model has been an alternative to simulating the effect of blockage, swirl, and suction due to fan blades with reasonable computing costs.[7–9] This approach uses body force terms to model flow turning and loss due to rotor/stator blade rows. The body force terms, extracted from single-passage Navier-Stokes flow simulation results or experimental test data, were added as source terms in the flow equations for grid cells swept by blade rows. The body force approach allows relatively accurate flow simulations of inlet-fan interaction problems without actual full-annulus simulation of the rotor/stator geometry. Xu[10] developed a viscous body force model, which extracted viscous body forces as source terms form unsteady Reynolds Averaged Navier-Stokes (RANS) solutions and directly solved Euler equations through blade passages. Chima[11] introduced a three-dimensional unsteady CFD code called CSTALL to solve the Euler equations through the entire annulus and all blade rows. And two computational fluid dynamics codes have been merged to permit rapid calculations of subsonic inlet-fan interaction.[7,12] However, It needed a third code called SYNCEX to handle data communication, storage, and synchronization. So, this method made it relatively complicated for the data transfer between CFD codes. In addition, it was also inconvenient to generate complicated geometry and mesh for subsonic inlet-fan calculations.

In the present study, a coupled supersonic inlet-fan Navier–Stokes simulation method was developed using COMSOL–CFD code. The COMSOL Multi-physics software environment is capable of facilitating all steps in the modeling and simulation processes from part defining, feature based meshing to visualization and solution analysis. The inlet and fan could been simulated simultaneously by different COMSOL modules, and the data transfer at each grid point of the inlet-fan interface was completed more easily by the form of boundary conditions than the SYNCEX code.[7,12] A three-dimensional body force model, in which viscous effects on the exchange of momentum between fluids and detailed viscous flow close to walls in blade passages can be calculated directly, was installed into the Navier–Stokes code of the COMSOL-CFD to simulate blade rows without specifying blade geometry. The governing equations for flow were written in non-conservative form in Cartesian coordinates with body forces as source terms on the right-hand side. Because the body force only changed the size of the mechanical energy with nothing on the size of the internal energy, the energy equation was written in the internal form. And coupled axisymmetric mixed compression supersonic inlet-fan simulations were conducted under Mach number 2.8 operating conditions, which were simulated by the High Mach Number Flow (HMNF) module of COMSOL Multi-physics.

The remainder of this paper is organized as follows. Section 2 describes formulation of the present body force approach. Section 3 presents numerical approaches for simulating flow, including flow solvers, mesh generation method, and boundary conditions. Section 4 presents validation results of the HMNF module for supersonic flows and the present body force model and coupled supersonic inlet-fan simulations. Finally, Section 5 provides the summary and conclusions.

2. Body force model

2.1. Governing equations

Based on the COMSOL-CFD code, the governing equations were written in non-conservative form in Cartesian coordinates with body forces as source terms on the right-hand side. And, viscous effects on the exchange of momentum between fluids and detailed viscous flow close to walls in blade passages were considered directly by viscous terms of governing equations as follows.

$$b\begin{bmatrix} \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{V}) \\ \rho \frac{\mathrm{D}V_x}{\mathrm{D}t} + \frac{\partial p}{\partial x} - \frac{\partial \tau_{xx}}{\partial x} - \frac{\partial \tau_{yx}}{\partial y} - \frac{\partial \tau_{zx}}{\partial z} \\ \rho \frac{\mathrm{D}V_y}{\mathrm{D}t} + \frac{\partial p}{\partial y} - \frac{\partial \tau_{xy}}{\partial x} - \frac{\partial \tau_{yy}}{\partial y} - \frac{\partial \tau_{zy}}{\partial z} \\ \rho \frac{\mathrm{D}V_z}{\mathrm{D}t} + \frac{\partial p}{\partial z} - \frac{\partial \tau_{xz}}{\partial x} - \frac{\partial \tau_{yz}}{\partial y} - \frac{\partial \tau_{zz}}{\partial z} \\ \rho \frac{\mathrm{D}e}{\mathrm{D}t} - M \end{bmatrix} = \boldsymbol{\Phi} \tag{1}$$

where $\begin{matrix} M = \frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right) + \frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right) - \\ \frac{\partial (V_x p)}{\partial x} - \frac{\partial (V_y p)}{\partial y} - \frac{\partial (V_z p)}{\partial z} + \frac{\partial (V_x \tau_{xx})}{\partial x} + \\ \frac{\partial (V_x \tau_{yx})}{\partial y} + \frac{\partial (V_x \tau_{zx})}{\partial z} + \frac{\partial (V_y \tau_{xy})}{\partial x} + \frac{\partial (V_y \tau_{yy})}{\partial y} + \\ \frac{\partial (V_y \tau_{zy})}{\partial z} + \frac{\partial (V_z \tau_{xz})}{\partial x} + \frac{\partial (V_z \tau_{yz})}{\partial y} + \frac{\partial (V_z \tau_{zz})}{\partial z} \end{matrix}$, t is time; x, y, z are the three directions of Cartesian coordinate; ρ is density; p is pressure; $\boldsymbol{V}$ is the velocity vector and $V_i (i = x, y, z)$ are the three components of velocity along x axis, y axis and z axis; $\tau_{ij} (i, j = x, y, z)$ are the nine components of shear stress in Cartesian coordinate; k is the thermal conductivity; e is total energy; T is temperature; b is the blockage factor; $\boldsymbol{\Phi}$ is the total body force.

Some of the variables in Eq. (1) are obtained by following formulas:

Please cite this article in press as: LI Q et al. Development of a coupled supersonic inlet-fan Navier–Stokes simulation method, *Chin J Aeronaut* (2017), https://doi.org/10.1016/j.cja.2017.11.011