Available online at www.sciencedirect.com

SCIENCE @ DIRECT®

European Journal of Mechanics B/Fluids 25 (2006) 379–392

Numerical simulation of unsteady blade row interactions induced by passing wakes

Pascale Kulisa *, Cédric Dano

LMFA UMR 5509, Ecole Centrale de Lyon, 36, av. Guy de Collongue, BP 163, 69134 Ecully cedex, France

Received 29 November 2004; received in revised form 7 April 2005; accepted 6 August 2005

Available online 19 December 2005

Abstract

A numerical study of the unsteady phenomena resulting of periodic passing wakes is presented. An unsteady passing wake boundary condition is implemented in a three-dimensional Navier–Stokes code. Unsteady computations are performed to evaluate the capability of the code to simulate the rotor–stator interaction flow. The analysis of the flow structures shows the vortical disturbances and the migration of the incoming wakes through the blade passage. This physical analysis allows to separate the main origins of the losses.

Keywords: Rotor–stator interaction; Numerical simulation; Wake

1. Introduction

In turbomachinery, the physical phenomena are numerous and complex. The fluid is strongly deviated and undergoes significant pressure and temperature variations. Many swirling structures appear and interact together. Performance improvement requires to load the blade by decreasing the blade number and to reduce the row spacing [1–3]. Interactions between adjacent blade rows are consequently increased. The aerodynamic blade load increase emphasises the separated zones appearance. Unsteady phenomena resulting from the periodic wakes due to the upstream blade row appear to be crucial, [4]. The turbulence modelling is also important to provide accurate numerical simulations. This fact is emphasised when heat transfer exists as it is the case for the turbine flows [5]. A RANS (Reynolds Averaged Navier–Stokes) approach is mainly used in turbomachinery because the geometry configurations are complex and the Reynolds number is high.

The present study evaluates the ability to reproduce the rotor–stator interactions phenomena without simulating the complete stage configuration and analyses the main origin of the losses in a turbine configuration. The computation is limited to a single stator passage. A boundary condition of passing wakes is prescribed to reproduce the unsteady effects of incoming wakes. The first part of the study consists in evaluating different two-equation turbulence models for turbomachinery applications. The objective is to select one model which is efficient for the blade wake prediction and sufficiently robust for the application area. Numerous steady computations were performed for turbine and

* Corresponding author. Fax: +33 (0)4 78 64 71 45.
E-mail address: pascale.kulisa@ec-lyon.fr (P. Kulisa).

0997-7546/$ – see front matter
doi:10.1016/j.euromechflu.2005.08.001

compressor test cases. The k–l model of Smith was retained. This first part was presented in details in [6]. The second part consists in simulating the rotor–stator interaction phenomena using an unsteady boundary condition. The present paper concerns this second part. The method is explained and we focus on flow structures occurring within the wake-blade interaction.

When the wakes of an upstream blade row progress through the downstream blade passage, characteristic pattern of vortical structure is created. The incoming wakes are cut and convected through the passage. A vortical flow is then produced at each side of the leading edge. Moreover, the wakes migrate towards the suction surface for a turbine (towards the pressure side for a compressor). The two vortical structures evolve differently on the suction side or on the pressure side of the blade.

On the pressure side, the incoming wake evolves into counter-rotating vortices. These vortices are the dominant source of disturbances over the blade pressure side. On the suction side, the flow is characterised by a vortical stream induced by the wake impact on the leading edge. The boundary layer on the suction side is distorted and aspirated from the surface.

Computations are performed with the Canari numerical code developed by Onera. This code solves the averaged three-dimensional Navier–Stokes compressible equations. The turbulence model used for this simulation is the low-Reynolds number two-equation model k–l developed by Smith [7]. A non-linear approach, avoiding the Boussinesq hypothesis, is used.

The steady computational procedure has been validated by experimental data comparisons on various turbine and compressor configurations [6]. In the present study, the wake propagation is evaluated in a high subsonic turbine cascade. The experimental conditions are close to those encountered in the real engine. Because the velocity level is high and the blade boundary layers are thin, detailed aerodynamic data in the blade passage are not available. The interest of this test case is to evaluate the ability of the method in complex and realistic turbine configuration. The stagnation pressure distribution downstream the blade is compared to measurements. Then the unsteady flow structures generated by the incoming moving wakes are analysed. For turbine cases, the interpretation of these unsteady effects has been discussed in [8–10]. We show that the physical phenomena taking place in the rotor–stator interaction are reproduced by this method. So this approach may be an efficient compromise between steady calculations and unsteady stage calculations.

2. Experimental configuration

The experimental configuration concerns the flow developing in a linear turbine cascade. The experimental study was carried out in the von Karman Institute piston compression tube facility [11], Fig. 3. The complete description of the wind tunnel is presented in [12]. This facility is made of three main parts: a 5 meter long and 1 meter diameter cylinder, the test section, and a downstream dump tank. The cylinder contains a lightweight piston driven by the air coming from a high pressure reservoir. As the piston is pushed forward, the gas located in front of it is isentropically compressed until it reaches the requested pressure and temperature levels. The valve is then opened, allowing the pressurised and heated gas to flow through the test section. The freestream conditions (total temperature, pressure, mass flow) are constant. The exit Mach number (about 0.8), Reynolds number and the freestream to wall temperature ratio can be adjusted independently. A steady flow condition is maintained during 500 ms. The total pressure is obtained by 3-hole probe mounted on a pneumatic traversing system located downstream of the blade. The measurements are performed at midspan. The pressure measurements present an uncertainty of ± 1 mbar. The heat flux uncertainty is $\pm 5\%$. To generate the incoming wakes, a rotating disc is set in front of the blade cascade. Cylinders are fixed on the disc diameter.

The main interest of this experiment is to provide realistic conditions near to the real engine. The price to pay is that only global quantities may be measured. So the present study allows to validate the approach in a complex situation.

3. Physical model and turbulence modelling

The physical model of the flow is based on the compressible three-dimensional Navier–Stokes equations. The governing system is composed by the conservation equations of the density, the momentum and the energy. These Reynolds averaged equations involves unknown correlations: the Reynolds stresses and the turbulent heat transfer. To close the energy equation, the turbulent thermal diffusivity is related to the eddy viscosity by a constant turbulent