Aerospace Science and Technology 78 (2018) 271–279

Contents lists available at ScienceDirect

Aerospace Science and Technology

www.elsevier.com/locate/aescte

Three-dimensional through-flow modelling of axial flow compressor rotating stall and surge

Mauro Righi, Vassilios Pachidis, László Könözsy*, Lucas Pawsey

Cranfield University, Cranfield, Bedfordshire, MK43 0AL, United Kingdom

ARTICLE INFO

Article history:
Received 8 December 2017
Received in revised form 27 March 2018
Accepted 12 April 2018
Available online 17 April 2018

Keywords:
Post-stall modelling
Through-flow code
Rotating stall
Surge
Axial compressors
Godunov-type schemes

ABSTRACT

This paper presents a three-dimensional through-flow approach based on the cylindrical Euler equations incorporating a body force method. Blade performance is captured through a mixture of empirical correlations and a novel reverse flow treatment. The code is the first application of a physically correct Godunov solver to three-dimensional rotating stall and surge modelling. This solver ensures the accurate calculation of inter-cell fluxes unlike in typical modern CFD codes in which the non-linear convective terms are linearised. Validation consists of modelling a low speed three-stage axial compressor in all operating regions, recreating the reverse flow, rotating stall and forward flow characteristics with good agreement to experimental data. Additional comparisons are made against rotating stall cell size and speed, to which good agreement is also shown. The paper ends with some full surge cycle simulations modifying both the tank volume after the compressor and the level of inlet distortion applied. Both tank volume and level of distortion have been found to affect the type of instability developed. The development of this code is a step forward in compressor rotating stall and reverse flow modelling and allows recreation of a full compressor map at a significantly low computational cost when compared to commercially available 3D CFD codes.

1. Introduction

The operation of a jet engine is limited by the onset of compressor stall. Rotating stall and surge are still not fully understood, neither in the mechanism through which the instability is triggered and subsequently develops, nor the aerodynamic load that is imposed on individual turbomachinery components during the event. This generally leads to a conservative approach during the design of blades and casings, and penalises the nominal operability range of the engine. Numerous solutions have been investigated to suppress the occurrence of compressor stall and increase surge margin. Experimental tests include axisymmetric arc-shaped slots casing treatment, investigated by Pan et al. [1], and bend skewed casing treatment tested by Alone et al. [2] Other recent methods proposed to increase the stability limit are recessed blade tips [3], casing grooves [4] and radial injectors [5]. Mhosen et al. [6] numerically investigated the use of tandem rotor blades to improve the flow turning and diffusion and suppress passage separation. Imani and Montazeri-Gh [7] proposed an improvement on the Min–Max limit protection in the control of aero-engines to reduce the possibility of accidental crossing of the stability limit. Aero-engines embedded in airframes are a common feature of next-generation aircrafts but the unsteady flow distortion generated by the S-ducts greatly reduce the surge margin. Gil-Prieto et al. [8], developed a method to predict peak levels of distortions in S-ducts in order to avoid extensive experimental testing and continuous redesign of the compressor when matching inlet duct and engine. The various post-stall behaviours that a multi-stage compressor can exhibit were demonstrated by Day and Freeman [9] on a Rolls-Royce VIPER compressor. As illustrated in Fig. 1, at high speeds the compressor is prone to surge, while at mid speed it forms a full span stall cell and at low speeds a front end stall.

Full characterisation of the post-stall behaviour of a high speed compressor experimentally is expensive. CFD modelling of axial turbomachinery in full 3D has been successfully achieved in the past, however the computational cost required is significant. In 1999 in a joint effort NASA and GE simulated the GE90 turbine system (18 blade rows) using 3D CFD [10]. The 9 million elements model was run on 121 processors in parallel and it took 15 hours to converge to a single operating point, in 10000 iterations. Gourdain et al. [11] simulated rotating stall using a quasi-3D code and modelling a stream surface; yet still a single stage of compression required 8 hours per revolution on a 32 core system. This reduced-order CFD was able to simulate a stable stall but the predicted

* Corresponding author.
E-mail address: laszlo.konozsy@cranfield.ac.uk (L. Könözsy).

https://doi.org/10.1016/j.ast.2018.04.021

Nomenclature

B	Greitzer's parameter	–
b	Blockage parameter	–
f	Volumetric force	N/m^3
W_{EX}	Work exchange	W/m^3
E_0, H	Internal energy, Enthalpy	J/m^3
E, F, G, S	Euler equations fluxes and source term	
M	Mach number	–
P	Pressure	Pa
R_{CUR}	Radius of curvature	m
r	Radius	m
T	Temperature	T
t, τ	Time, time constant	s
u	Air velocity	m/s
U	Blade velocity	m/s
U	Conservative variables	
x	Axial position	m
α	Air flow angle	rad
β	Camber-line direction	rad
γ	Stagger angle	rad
γ	Specific heat ratio	–
δ	Deviation	rad
θ	Circumferential position	rad
ρ	Density	kg/m^3
Ψ	Total-to-total pressure rise coefficient	–
ϕ	Flow coefficient	–
Ω	Rotational speed	rad/s
ω	Pressure loss coefficient	–

Subscripts

r, x, θ	Radial, axial, circumferential direction
η, ξ	Parallel, normal to camber-line direction
0	Total quantity
SS	Steady-state value

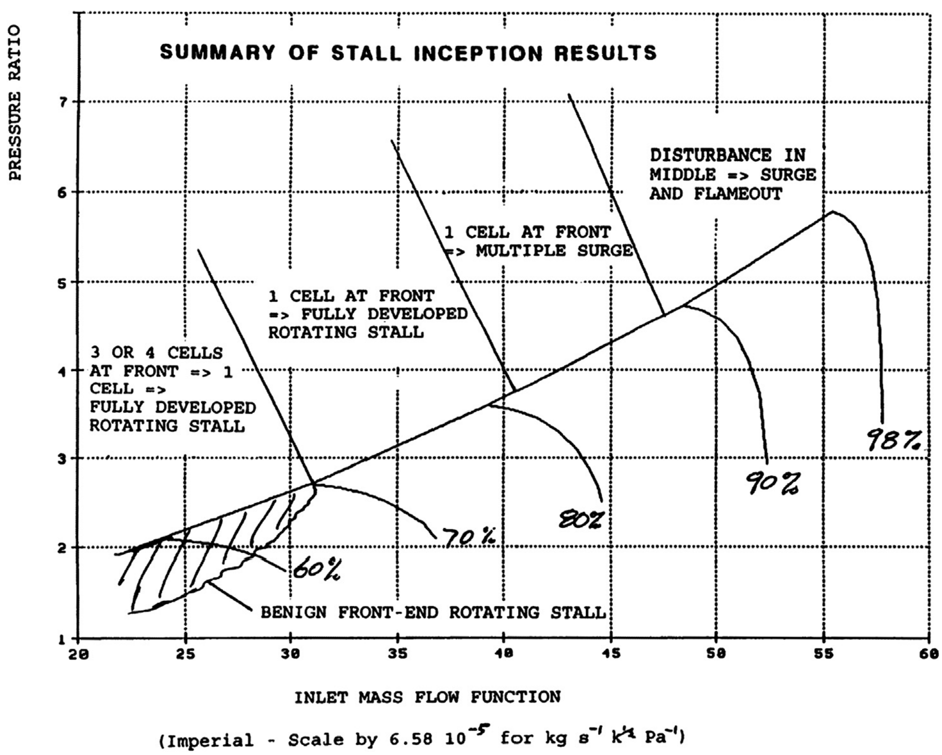

Fig. 1. Stall inception behaviour of Rolls-Royce VIPER compressor as simulated by Wilson [15].

number and speed of the cells was wrong. Khaleghi modelled rotating stall inception on a half annulus [12,13]. While the model had a span resolution at a low computational cost, this technique was able to simulate only modes multiple of two. More recently a URANS-based CFD method has been used to model stall in a modern high speed compressor by Dodds [14]. Modelling only the first three stages of the original eight, using the full annulus, one shaft revolution took 48 hours of computational time on 64 CPUs. An alternative to such an approach is the use of a through-flow code, which involves simplifying the problem using coarse grids and flow modelling based on the Euler equations with blade performance taken into account by empirical correlations. Such codes are extensively used in industry, with 2D codes looking only at the meridional plane and 3D codes modelling the full annulus, available for studying conventional pre-stall compressor performance. Some codes have been extended and adapted to simulate reverse flow and stalled flow, an example of which is Wilson [15] using a series of 1D through-flow codes in parallel to model the various stalling behaviours of the VIPER engine (as shown in Fig. 1). Gong [16] used a body force method, based on a database of correlations from conventional CFD, to model rotating stall. The tool was developed by a number of other researchers within MIT which improved the method, allowing it to be applied to a multi-stage axial compressor, as reported by Brand [17]. Longley [18] proposed a novel method to predict the region of highly separated flow which occurs during compressor stall. Since the circumferential element density is not enough to capture the separated flow between blade rows, Longley introduced a variable b that captures the flow non-uniformity. An equation is added that permits the creation, transport and mixing of the non-uniformity that represents the physics