Computers & Fluids 116 (2015) 192–204

Contents lists available at ScienceDirect

Computers & Fluids

journal homepage: www.elsevier.com/locate/compfluid

URANS analysis of the effect of realistic inlet distortions on the stall inception of a centrifugal compressor

S. Vagnoli *, T. Verstraete

Von Karman Institute for Fluid Dynamics, Chaussee de Waterloo 72, 1640 Rhode Saint Genese, Belgium

ARTICLE INFO

Article history:
Received 13 August 2014
Received in revised form 21 January 2015
Accepted 21 March 2015
Available online 16 April 2015

Keywords:
Radial compressor
Stall
URANS
Inlet distortion

ABSTRACT

In this paper unsteady RANS computations are used to study the inception of rotating stall in a transonic centrifugal compressor taking into account realistic installation effects on performance, as commonly found nowadays due to space limitations. To this end, the effect of an ideal uniform inlet conditions (normally found for long straight inlet) is compared with inlet distortions generated by a bent pipe installed just in front of the impeller.

The numerical techniques that have to be applied to correctly represent the rotating stall are explained in detail: all simulations are done modeling the whole annulus of the radial machine, using high performance computing to represent the non-periodic phenomena leading to the stall inception. Moreover, stable boundary conditions are employed, with the aim to avoid the inception of large unphysical surge cycles.

When an uniform inlet flow to the compressor is considered, the formation of 8 blockage cells rotating in the same direction of the compressor is pointed out. On the other hand, when the elbow is installed in front of the impeller, the distorted flow suppresses the formation of a rotating stall pattern.

1. Introduction

Industrial applications of radial compressors demand increasingly higher operating ranges. To this end, more accurate predictions of the stall limit are required, as well as a better understanding of the mechanisms leading to the transition to stall. This allows for the implementation of technical solutions which can enhance the stability of compressors.

Today the design of efficient and stable compression systems becomes increasingly more challenging considering that more compact solutions are required, resulting from strict limits on the encumbrance and weight. Because of this, bent pipes (elbows) are often installed in the proximity of the compressor inlet, forcing the impeller to operate with a nonuniform inlet flow, which will affect the performance and the stall limit.

The spike stall inception [1] is a progressive phenomenon beginning with a flow separation in one particular blade, usually limited to the tip region, which leads to the development of one or more rotating stall cells.

For centrifugal compressors, it is not as trivial to identify the occurrence of stall as in axial compressors. This is particularly due to the fact that in radial impellers the presence of rotating stall cells is often indicated by an increase of noise only, without an abrupt change of performance [2], since the centrifugal effect on compression is always predominant on the aerodynamic one [3].

For unshrouded transonic compressors, one of the most important initiators of rotating stall is the interaction between the tip clearance vortex and the main flow [4–6], which lead to the first inception of rotating disturbances within the machine [7,8]. It is generally agreed that at the stall limit the interface between incoming and tip flow lines up with the leading edge of the blades [5]. Based on that, methods to increase the stall limit by means of casing treatment of the shroud have been often proposed and analyzed [9].

Because of the intrinsically time-dependent nature of the stall itself, it is not trivial to reproduce the rotating stall inception by means of CFD: an unsteady approach shall be used, and the whole annulus of the compressor has to be analyzed, to take into account the non-periodic time-dependent nature of the phenomenon. Such kind of approach is highly time consuming, and has become possible only in recent years, thanks to an increased computational power.

Few studies can be found in the literature with the aim to numerically reproduce the rotating stall, mainly related to axial

* Corresponding author.
E-mail address: stefano.vagnoli@vki.ac.be (S. Vagnoli).
URL: http://www.vki.ac.be (S. Vagnoli).

http://dx.doi.org/10.1016/j.compfluid.2015.03.015
0045-7930/

Nomenclature

Mred	reduced mass flow
Ma	Mach number
N_{cells}	number of stall cells
$\dot{m}$	mass flow
c	Chord length at 50% span
f	frequency
p	pressure
r	radius
t	time
z	axial direction
Greek symbols	
β	pressure ratio
δ	geometric angle at the pipe outlet
η	effectiveness
τ_{rot}	time over time of a rotor revolution
ω	rotation velocity
Subscripts	
0	referred to ambient
imp	impeller
main	main blade
rel	relative frame of reference
rot	rotations
tip	referred to tip
TT	total to – total

compressors [4,10–12]. This emphasizes a lack of similar studies applied to centrifugal compressor.

Chen et al. [4] and Gourdain et al. [12] compare numerical results with experiments for axial compressors in stall. A close prevision of the operation point at the stall inception limit between CFD and experiments is pointed out by both authors.

An element that could heavily influence the rotating stall inception is the non-uniformity of the flow entering the compressor. This can be due to for example bent pipes installed close to the inlet, because of strict space constraints to be taken into account during the design of many applications.

For radial compressors, Engeda et al. indicated a lack in literature about inlet distortion effects [13]. Ariga et al. [14] studied the influence of different types of total pressure inlet distortions experimentally, comparing the performance of the impeller with distorted and undistorted inlet flow. The degradation of performance due to nonuniformities becomes more remarkable at high mass flow, while little effect is noticed at lower mass flow, as also confirmed by other authors [15].

From a numerical point of view, Kim et al. [16] investigate three different bends with the aim to reduce the nonuniformity at the impeller inlet and to improve the performance of the machine. In this case, steady CFD is only used to design the inlet duct, but the effect on the radial compressor itself is not investigated.

The aim of the current paper is to analyze the transition to stall of a transonic centrifugal compressor in detail through numerical simulations, discussing an approach that is physically accurate and stable. Two configurations with respectively a straight inlet and an elbow are investigated by means of unsteady RANS simulations, reproducing the non-periodic nature of the rotating cells by full annulus models of the machine.

2. The flow solver and numerical model

2.1. Analyzed geometry

The geometry used in this investigation is shown in Fig. 1(a). It is a transonic centrifugal compressor, whose shape has been optimized at the von Karman Institute for Fluid Dynamics. The geometry consists of an unshrouded impeller with 9 main and 9 splitter blades, and a vaneless diffuser. The main design parameters are shown in Table 1.

Two different configurations for the flow at the inlet have been considered. In the first case, a straight pipe is modeled, whilst in the second case a 90° bent section is imposed upstream of the previous domain. Dimensions of the bent pipe are shown in Fig. 1(b).

No experimental data are available for the geometry considered, therefore the validation of the numerical approach was performed on the NASA rotor 37 [17]. This test case was chosen since it presents a very similar behavior to the aforementioned centrifugal impeller at near stall conditions. The validation is presented in Session 3.

2.2. The flow solver

In the present work, the commercial software FINE/TURBO has been used to solve both steady and unsteady compressible Reynolds-Averaged Navier–Stokes (URANS) equations. This is a second order cell-centered finite volume solver for structured multiblock meshes. The code was already successfully employed in different circumstances to represent unsteady flow patterns of tip loaded transonic axial and radial compressors operating at near stall conditions [18–20].

For the unsteady RANS simulations 405 time steps are used to model one rotation of the compressor. The solution for every time step is obtained by means of 100 internal iterations, using the 4th order Jameson's scheme. A maximum Courant number of 2 is imposed for the inner iterations. The turbulence model employed is the Spalart–Allmaras one equation model.

2.3. Boundary conditions for the unsteady analysis

When numerical simulations at near stall and stall conditions have to be performed, the choice of proper outlet boundary conditions needs to be discussed in detail.

To induce stall in the compressor in fact, the machine has to operate at low mass flow. In this state, the system is also more prone to surge.

Surge is a dynamic resonance of the whole compression system (of which the compressor is only a part), leading to low frequency and high amplitude fluctuations of the mass flow. Since the whole system is directly involved, surge can only be reproduced by directly including the resisting circuit in the numerical model, with a consequent strong increase of the computational cost. Therefore, when the analyzed domain represents only the compressor, the inception of surge has to be avoided to prevent unphysical fluctuations of the mass flow.

Greitzer's theory [21] identifies two different limits for surge inception: