

AXIAL TRANSONIC COMPRESSOR STALL MECHANISM WITH DIFFERENT ROTATIONAL SPEEDS

NASEEM AHMAD^{1,2*}, QUN ZHENG², LI HEFEI²,
GHULAM ISHAQUE²

¹*Department of Mechanical Engineering, Institute of Space Technology,
Islamabad 44000, Pakistan*

²*College of Power and Energy Engineering, Harbin Engineering University,
Harbin 150001, China*

[Received: 26 October 2020. Accepted: 05 April 2022]

doi: <https://doi.org/10.55787/jtams.22.52.2.144>

ABSTRACT: In the current research work multi passage, steady RANS, and URANS simulations were carried out on NASA rotor 37 to investigate the stall mechanism of the axial transonic compressor with three rotational speeds. 100%, 80%, and 60% rotational speeds were selected to examine the behavior of the rotating stall. The basic flow mechanism was achieved by examining the blade passage field thoroughly. The interaction between the shockwave and TLV produced blockage, which led to rotating stall for 100% and 80% rotational speeds. The spilled flow at the leading edge, which was caused by the TLV was the main reason for the rotating stall at 60% of rotational speed.

KEY WORDS: axial compressor, numerical simulation, stall mechanism, three rotational speeds, unsteadiness.

NOMENCLATURE

URANS : unsteady Reynolds averaged Navier Stokes	
TLV : tip leakage vortex	TCF : tip clearance flow
LE : leading edge	TE : trailing edge
TLF : tip leakage flow	CFD : computational fluid dynamics
SST : shear stress transport	SS : suction surface
PS : pressure surface	BPF : blade passing frequency
y^+ : non-dimensional wall distance	BLS : boundary layer separation

*Corresponding author e-mail: naseem.saddiqui@yahoo.com

1 INTRODUCTION

The axial compressor is an important and key component of gas turbines, such as the jet engine, aerospace engine, and marine engine. Substantial improvements have been done on the aerodynamic performance model, design optimization technique, and mechanism of flow loss for the axial compressor. The accuracy of the acceptable level at design mass flow rate could be achieved. But axial compressors at low mass flow rate underwent aerodynamic instabilities like rotating stall and surge.

Inoue et al. [1] worked experimentally on the axial compressor with different tip gaps to investigate the behavior of the TLV. They reported that increasing the tip gap increased the TLV and further explained that a large tip gap deteriorates the efficiency and performance of the compressor.

TLV is the main factor of the compressor instability near the casing zone. Day and Freeman [2] worked on an eight-stage compressor and compared the results with low-speed laboratory data. They confirmed that low-speed technique is effective means of gaining the behavior of the high-speed compressor. TLV breakdown at the design point near the stable region of the compressor rotor was investigated by Schlechtriem and Ltzerich [3]. According to their report, significant blockage occurred in the flow field due to back pressure which led to a rotating stall. 3D RANS computations were made by Chima [4] for tip clearance flow in an axial compressor. He elucidated that the clearance vortex and passage shock were inviscid phenomena and further elaborated that shock vortex interaction was followed by high rotational flow with a large separated zone close to the stall point. Four high-speed axial compressors were tested by Day et al. [5] to examine the stall inception. They reported that spike type stall inception was featured at low speed and modal type stall was raised at the mid-speed range.

A numerical study was conducted by Hofmann and Ballmann [6] to investigate the development of tip clearance vortex and vortex shock interaction in an axial transonic compressor. They reported that tip vortex advanced to passage shock and the flow vortex decelerated and abrupt changes occurred in the vorticity distribution. Yamada et al. [7] investigated the unsteady flow field of transonic compressor rotor 37. The simulation results confirmed that TLV breakdown arose in the compressor due to vortex interaction with the passage of the shock wave. Bubble-type TLV breakdown happened at peak condition, while spiral type breakdown appeared due to shock wave and vortex interaction close to stall condition. Yamada et al. [7] worked on rotor 37 to investigate the behavior of TCF by performing multi-passage simulations. They revealed that TLV breakdown traveled up from downstream due to shock wave near stall limit due to reduced mass flow rate. Furthermore, TLV fluctuations went downstream and interacted with the LE of the adjacent blade.

Biollo and Benini [8] modeled a transonic novel compressor rotor started from NASA rotor 37. They concluded that the new rotor showed minimum detrimental shock boundary interaction at low flow operational conditions. Rotor 37 was numerically investigated by Zhang et al. [9] to understand the mechanism of unsteady TLF and the inception of the rotating stall. They reported that the TLV breakdown happened after the blade passage shock. The breakdown of the unsteady TLV gave TLF blocking and it spilled at the LE of the rotor blade. Li et al. [10] conducted an experimental investigation on the transonic compressor to examine the instability at different speeds. According to their experimental investigation, a disturbance of low frequency was initiated at the rotor hub. This disturbance was equal to the Helmholtz frequency and it was constant through the instability inception. A numerical method was applied by Zhang et al. [11] to investigate the stall mechanism in an axial compressor. They revealed that TLV was caused by the tip blockage which leads to a rotating stall. The conclusion demonstrated that rotating stall occurred due to long-scale disturbance.

Wu et al. [12] investigated numerically the flow field of the transonic compressor near the casing region. The CFD results demonstrated that various tip vortices were developed at peak efficiency conditions due to the shockwave. But at low mass flow rate, TLV was produced, and further, when mass flow rate was decreased close to stall point, shock wave interacted with TLV which caused a stall in an axial compressor. Multipassage CFD simulations on rotor 37 were conducted by Lang et al. [13] to examine the behavior of rotating stall. They explained that TLV breakdown happened due to detached shockwave and when the blockage zone was stronger than the TLF and vice versa. Moreover, the area of the blockage changed periodically due to the oscillation of the flow.

Shubo Ye et al. [14] worked on the casing grooves to assess the rotating instability for the design and part speed. They revealed that at partial speed the circumferential grooves reduced the blockage of two types, local vortex, and suppressed vortex, at the SS of the blade. While at the design speed the circumferential grooves minimized the amplitude of the fluctuation of flow near the stall limit. A parallel CFD technique was developed by Liu et al. [15] to investigate the rotating stall and surge phenomenon in rotor 35. They reported that the aerodynamics instability itself manifested as a rotating stall in the convergence of unsteady performance. Also, the stall cells and rotating speed varied with the stall inception.

Smith et al. [16] investigated the tip leakage flow by conducting unsteady multi passage simulations. They revealed that the blockage of the corner separation at zero tip gap was the leading mechanism. When the tip gap has increased the flow, leakage reduced the blockage and the range of the flow was increased. Weixiong et al. [17] studied the unsteady disturbance and TLV during the process of pre-stall

in the contra-rotating compressor. One passage of unsteady simulation was carried out. The CFD results explained that unsteady fluctuation appeared at the mass flow coefficient of 0.3622 and frequency of 2.86 BPF. The phenomenon of stall point is indicated in downstream of the compressor rotor. Furthermore, the position of the maximum fluctuation was mainly dependent on the mainstream and TLV interface at the downstream of the compressor rotor.

The mechanism of the stall of NASA rotor 37 at 100% design speed has been investigated by numerous researchers, while studies of the mechanism of the rotating stall at partial speeds are rare. So, unsteady numerical simulation was carried out in this research for NASA rotor 37 at 60%, 80%, and 100% rotational speeds to investigate the mechanism of the stall. Also, to provide the theoretical background for the various rotational speeds to improve the stability and performance of the axial compressor by using 3D URANS equations.

2 NASA ROTOR 37 DESIGN SPECIFICATIONS

NASA rotor 37 is selected for this research. Originally the rotor was designed as an inlet rotor for the core compressor and in late 1970 the compressor was tested at the NASA Research Centre. The design specifications of the compressor rotor are given in Table 1. The design pressure ratio of the rotor is 2.106 at 20.19 kg/s mass flow rate. The rotor has 36 blades with a 0.7 hub to tip ratio and the aspect ratio of the rotor is 1.19. The tip gap is 0.356 mm and the rotation speed of the rotor is 17,188.7 rpm, which is taken from Dunham, AGARD report (1998) Total pressure and total

Table 1: Design specification of NASA rotor 37 [18]

Design specification	Value
Number of rotor blades	36
Total pressure ratio	2.106
Rotor tip solidity	1.29
Design mass flow rate (kg/s)	20.19
Rotational speed (rpm)	1,7188.7
Rotor tip relative inlet Mach number	1.48
Rotor inlet hub-to-tip diameter ratio	0.7
Rotor hub relative inlet Mach number	1.13
Mass flow choking (kg/s)	20.93
Reference temperature (K)	288.15
Tip clearance (mm)	0.356
Tip speed (m/s)	454.136
Rotor polytropic efficiency	0.889

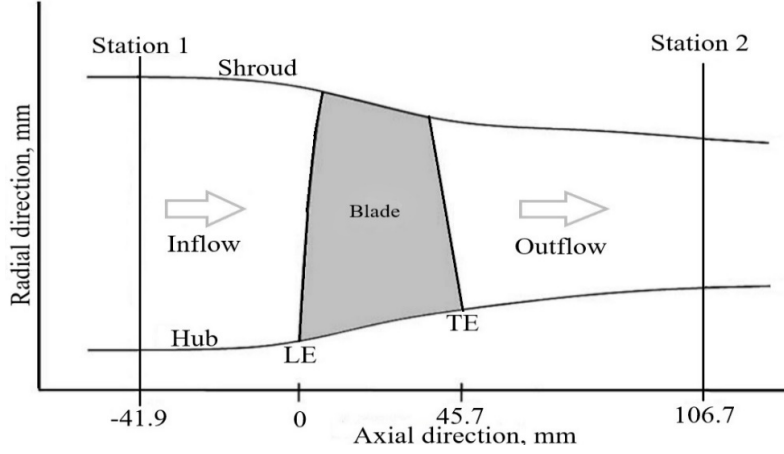


Fig. 1: Meridional view of NASA rotor 37.

temperature are measured at the inlet (station 1) and this is related to outlet mass flow rate (station 2). The distance between station 1 and blade LE is -41.9 mm, whereas station 2 is situated 106.7 mm downstream from the TE of the blade near to hub. The meridional plane of NASA rotor 37 is shown in Fig. 1.

3 GOVERNING EQUATIONS AND NUMERICAL METHOD

The SST turbulence model is used in numerical simulations. The SST model based on k - ω accounts for the transport equation of the turbulent shear stress and gives extremely precise predictions of the onset and the quantity of the separation flow under adverse pressure gradients. The appropriate transport behavior can be got by a limiter to the eddy-viscosity formulation

$$(1) \quad \mu_t = \frac{a_1 \rho k}{\max(a_1 \omega, S F_2)},$$

where

$$(2) \quad v_t = \frac{\mu_t}{\rho}.$$

F_2 is a blending function analogous to F_1 , that confines the limiter to the boundary layer wall, as the underlying supposition is not right for the free shear flows. S is the measure of an invariant for the strain rate. The SST k - ω model for the transport equations

$$(3) \quad \frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k U_i)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[(\mu + \mu_t \sigma_k) \frac{\partial k}{\partial x_i} \right] + \widetilde{P}_k - \beta^* \rho k \omega,$$

$$(4) \quad \frac{\partial(\rho\omega)}{\partial t} + \frac{\partial}{\partial x_i}(\rho\omega U_i) = \frac{\partial}{\partial x_i} \left[(\mu + \mu_t \sigma_\omega) \frac{\partial \omega}{\partial x_i} \right] + \alpha \rho S^2 - \beta \rho \omega^2 + 2(1 - F_1) \rho \sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i}.$$

F_1 , the blending function, is explained by

$$(5) \quad F_1 = \tanh \left\{ \left\{ \min \left[\max \left(\frac{\sqrt{k}}{\beta^* \omega y}, \frac{500\nu}{y^2 \omega} \right), \frac{4\rho \sigma_{\omega 2} k}{CD_{k\omega} y^2} \right] \right\}^4 \right\}$$

with $CD_{k\omega} = \max \left(2\rho \sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i}, 10^{-10} \right)$ and y is the distance to the closest wall away from the surface (k - ε model), F_1 is equal to zero and changes over to inside the boundary layer

$$(6) \quad F_2 = \tanh \left[\left[\max \left(\frac{\sqrt[2]{k}}{\beta^* \omega y}, \frac{500\nu}{y^2 \omega} \right) \right]^2 \right].$$

In the shear stress transport (SST) model a limiter of production is used to avoid the turbulence build-up in the stagnation zones

$$(7) \quad P_k = \mu_T \frac{\partial U_i}{\partial x_j} \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \rightarrow \tilde{P}_k = \min(P_k, 10 \cdot \beta^* \rho k \omega).$$

All the constants are calculated by a mixture from the respective constants of the k - ε and the k - ω model through $\alpha = \alpha_1 F + \alpha_2(1 - F)$. For this model the constants are:

$$\begin{aligned} \beta^* &= 0.09, & \alpha_1 &= 5/9, & \beta_1 &= 3/40, \\ \alpha_2 &= 0.44, & \beta_2 &= 0.0828, & \sigma_{k_1} &= 0.85, \\ \sigma_{k_2} &= 1, & \sigma_{\omega_1} &= 0.5, & \sigma_{\omega_2} &= 0.856. \end{aligned}$$

The rotor was studied at three different types of rotational speeds using a periodic blade passage. Multiblock structure grids were used which were generated by Auto-grad5 NUMECA [19]. The mesh computation is depicted in Fig. 2. The domain of the computational grid was split into two regions, the main flow region, and the tip gap region. O type grids were constructed at the blade LE and TE, which consisted of 65 Cells in the spanwise direction, 57 cells in the pitch-wise direction, and 65 cells in the chordwise direction. The main flow region was constructed with H-type grids.

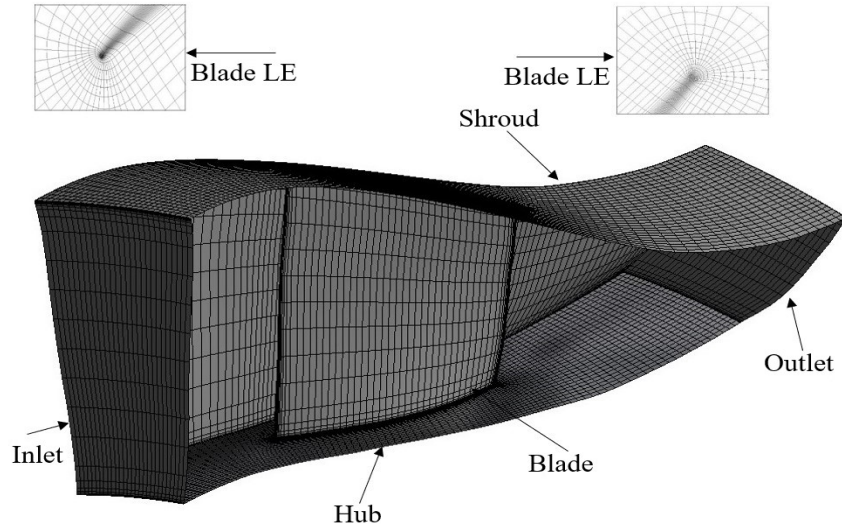


Fig. 2: Computational mesh of rotor 37.

The minimum grid spacing was set to 3.0×10^{-6} m to capture $y^+ < 2$ at the surface of solid walls. An inlet block was constructed with $17 \times 17 \times 21$ grids and an outlet block was constructed with $17 \times 17 \times 25$. The tip gap was constructed with $17 \times 21 \times 33$. The grid points of 480,000 were selected for this research. The commercial solver flow ANSYS CFX 17 [20] was used for all simulations. The results were achieved by solving URANS 3D equations and the equations were discretized using the finite volume method. The validation for the adiabatic efficiency and total pressure ratio was taken from the author's previous research Naseem et al. [21] and this is shown in Fig. 3.

The adiabatic efficiency and total pressure agree well with the experimental data. Multi-passage (three passage) simulations were performed by imposing periodic boundary conditions along with the circumferential direction boundaries. SST turbulence model was adapted for all the numerical simulations. The total pressure, total temperature, and flow angle were specified at the inlet with a turbulence intensity of 5%. Adiabatic with no-slip wall and smooth wall for the compressor rotor blade were selected in boundary details. The radial equilibrium state was fixed at the outlet. Backpressure at the outlet was increased by fixing the radial equilibrium state to analyze static pressure radial distribution. Regarding simulation at part speeds, backpressure was increased at the outlet. Similar convergence criteria are used which was proposed by Chen et al. [22] in the current research work. This criterion states that for three hundred steps mass flow rate difference at the inlet was less than 0.001 kg/s.

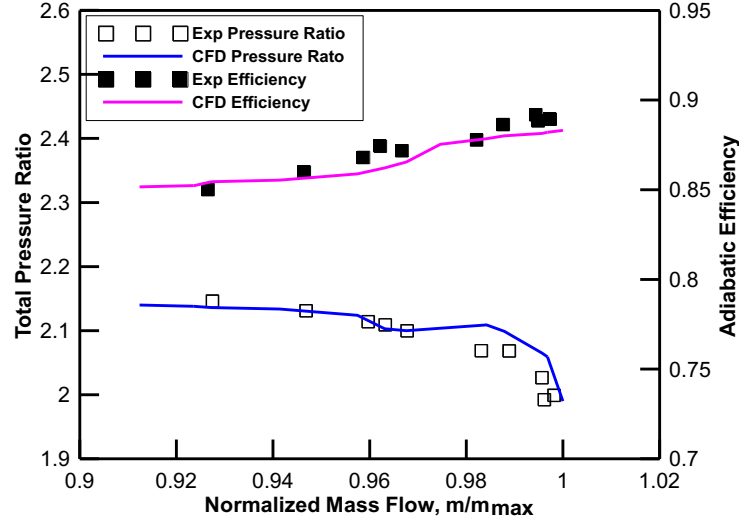


Fig. 3: Validation of adiabatic efficiency and total pressure ratio for NASA rotor 37 Naseem et al. [21].

The inlet and outlet mass flow rate variation were less than 0.5%. The variation of adiabatic efficiency per hundred steps was less than 0.3%. In this research work, the

Table 2: Values of y^+ for different surfaces

Surfaces		222,000 Grids	310,000 Grids	390,000 Grids	480,000 Grids	685,000 Grids
Hub	$y_{\min}^+$	0.18	0.13	0.11	0.08	0.05
	y_{ave}^+	4.4	3.15	2.47	2.01	1.42
	$y_{\max}^+$	8.01	5.67	4.55	3.70	2.51
Shroud	$y_{\min}^+$	0.94	0.61	0.51	0.40	0.29
	y_{ave}^+	3.52	2.47	1.99	1.59	1.52
	$y_{\max}^+$	7.36	5.23	4.09	3.77	1.76
Pressure side	$y_{\min}^+$	0.24	0.22	0.19	0.13	0.05
	y_{ave}^+	5.57	3.51	3.09	1.84	0.97
	$y_{\max}^+$	10.95	6.53	5.98	4.33	1.73
Suction side	$y_{\min}^+$	0.23	0.14	0.11	0.07	0.03
	y_{ave}^+	4.41	3.07	2.41	1.68	0.84
	$y_{\max}^+$	8.14	5.52	4.41	3.67	1.52

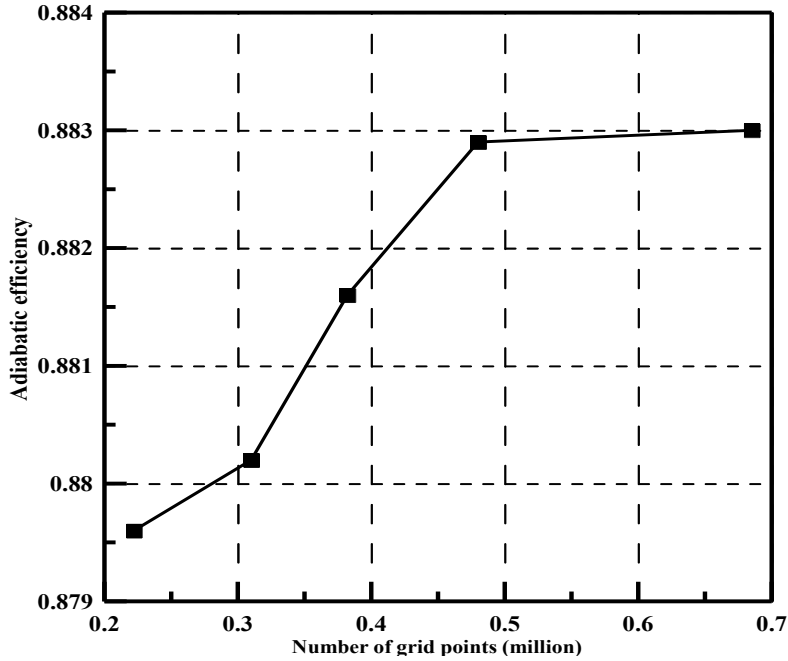


Fig. 4: Simulation results with a different number of grids.

same convergence criterion was used numerically for the initiation of the stall. To eradicate the influence of the grid numbers on the flow field, a grid independence test was performed for rotor 37. Table 2 summarized minimum, average and maximum values of y^+ for the hub, shroud, pressure side and suction side of the compressor rotor. Total pressure and total temperature of 101,325 Pa and 288.15 K were set at the inlet. Static pressure was changed at the outlet. A total of five different cases for the grid numbers were calculated. The calculated rotor adiabatic efficiency is depicted in Fig. 4. For the adiabatic efficiency, there is very little change from 0.48 million to 0.685 million. Given the computational cost and accuracy, grid points of 0.48 million were selected for this research work and the total number of grid points for three passages are 1.44×10^6 million.

4 RESULTS AND DISCUSSION

Three different rotational speeds were investigated for the stall mechanism. The simulations and experimental performance for the three rotational speeds are depicted in Fig. 5. The performance of the compressor rotor was calculated using one passage steady simulations. The experimental data for the 60%, 80%, and 100%, were taken

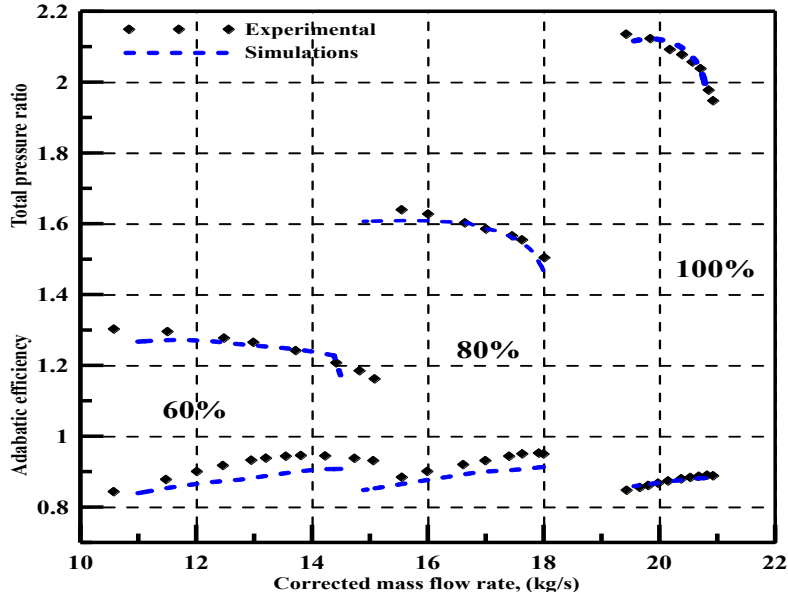


Fig. 5: Compressor map for three rotational speeds.

from Suder et al. [23]. For the design speed (100%) the experimental and simulated mass flow rate at choked conditions was 20.93 kg/s and 20.77 kg/s respectively. The difference between the experimental data and CFD simulations at choked mass flow rate was 0.16%. Moreover, the mass flow rate at near stall conditions for the simulations and the experiment was 19.52 kg/s and 19.42 kg/s correspondingly. The difference between both calculations was 0.10%. So, these results demonstrate that the stall and choked mass flow rate were predicted very well with the CFD simulation technique. Figure 5 depicted the adiabatic efficiency and total pressure ratio for 60%, 80%, and 100% respectively. For the design speed (100%) the total pressure ratio of the CFD calculation was smaller than the experiment but the adiabatic efficiency matched well with the experiment.

For 80% rotational speed, the total pressure ratio of the CFD simulation was lower than the experiment but it gave a larger stall margin. The adiabatic efficiency for CFD simulations showed a small discrepancy with the experiment. For the 60% rotational speed, the pressure ratio was smaller than the experiment. Also, the adiabatic efficiency was lower than the experiment. The difference between the adiabatic efficiency for 100%, 80%, and 60% was 0.58%, 3.03%, and 1.26% respectively. The difference in the adiabatic efficiency for the experiment and CFD simulation is caused by many aspects, for example, the real geometry of the blade, different turbulence

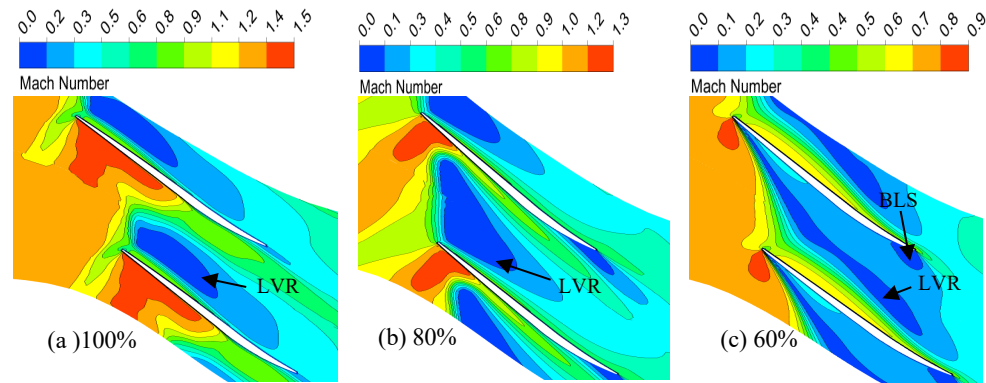


Fig. 6: Mach number contours for three rotational speeds near stall conditions.

models, etc. Overall, the performance of the CFD simulation matches pretty well with the experimental data. With the above knowledge, the stall mechanism of the compressor rotor for three different rotational speeds was investigated. The flow field was analyzed for three different rotational speeds (60%, 80%, and 100%) because the boundary conditions for the three rotational speeds near the stall point were very close to the experimental data.

Mach number contours at 98% for the three rotational speeds near stall conditions are depicted in Fig. 6. The variation of the TLV and shockwave for the three rotational speeds are discussed in detail. At 100% rotational speed, the fluid of low energy turned up downstream with the shockwave which influenced the blockage. The low energy fluid zone extended dramatically and the shockwave in the rotor passage was bowed near stall condition. The shockwave interacted with the TLV near the LE of the blade. At 80% rotational speed, the shockwave pushed down from the leading edge of the blade. The trajectory of the TLV along with a high region of low velocity turned up in the rotor passage. At 60% rotational speed, the low-velocity flow showed different behavior. Firstly, TLV covered a large area of low velocity. The second was the boundary layer flow separation at the suction surface of the blade trailing edge.

Figure 7 depicts the streamwise distribution of the pressure at 98% span for all three rotational speeds near the stall point. The pressure difference close to the leading edge of the blade was higher for 100% and 80% rotational speeds with the throttling of the compressor. TLV was stronger in this region due to greater pressure differences. The position of the shockwave was different for both 100% and 80% rotational speeds and the shockwave difference appeared due to the pressure differences. At 80% rotational speed the pressure was low that is why the shockwave shows different behavior. The other reason was that at 80% rotational speed, the TLV devi-

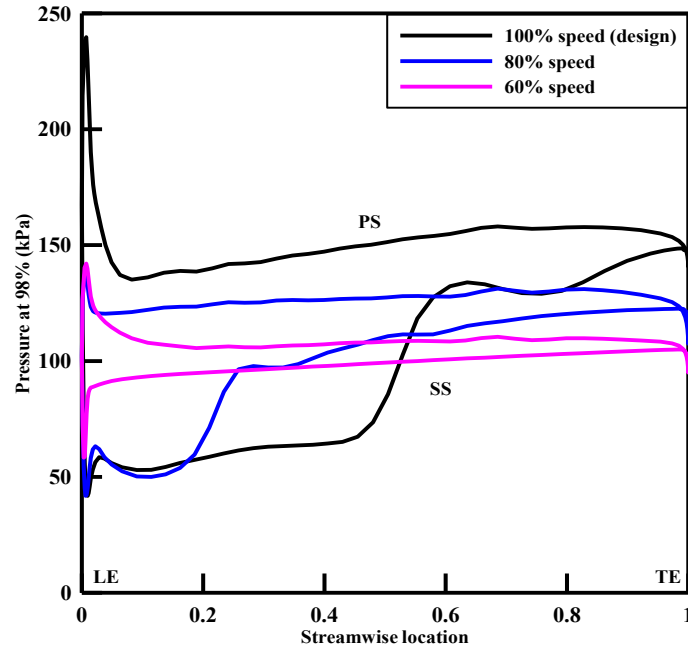


Fig. 7: Distribution of streamwise pressure at 98% span for the three rotational speeds near stall condition.

ation angle was larger than at 100% rotational speed. The shockwave demonstrated closer proximity to the leading edge at 80% rotational speed. For 60% rotational speed, no shockwave appeared due to very low pressure and low fluid velocity.

Surface streamlines near stall conditions on the blade suction surface for the three rotational speeds are depicted in Fig. 8. At 100% rotational speed, the interaction between the shockwave and tip leakage vortex can be seen in Fig. 8(a) close to the blade tip at the mid chord of the blade. With the decreased mass flow rate. The shockwave advanced upstream due to separation that occurred close to the blade tip. At 80% rotational speed, the interaction between the TLV and the shockwave could be observed in the middle of the blade suction surface. The separation can also be seen in the trailing edge of the blade. It can be concluded that the reduction of the rotational speed increased the separation further from TLV. At 60% rotational speed, no shockwave was observed. Instead of a shockwave boundary layer separation flow appeared and it can be seen on the suction surface of the blade near the trailing edge. The streamlines show different behavior for the 60% rotational speed near the stall limit due to low fluid velocity and pressure.

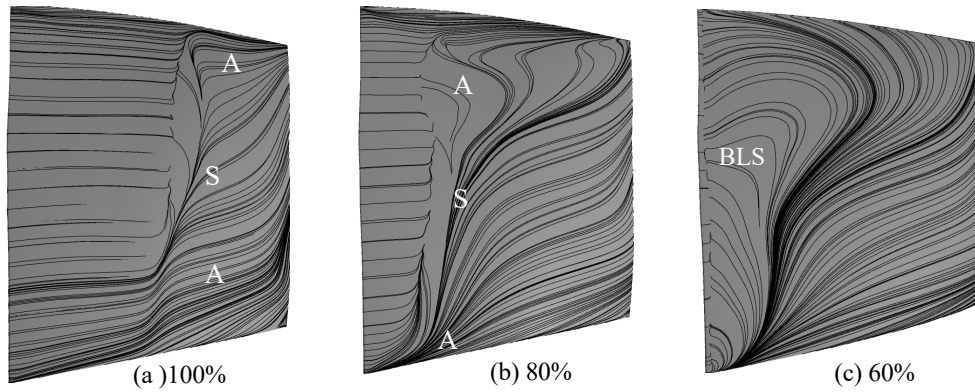


Fig. 8: Surface streamlines near to stall condition on the blade suction surface for three rotational speeds.

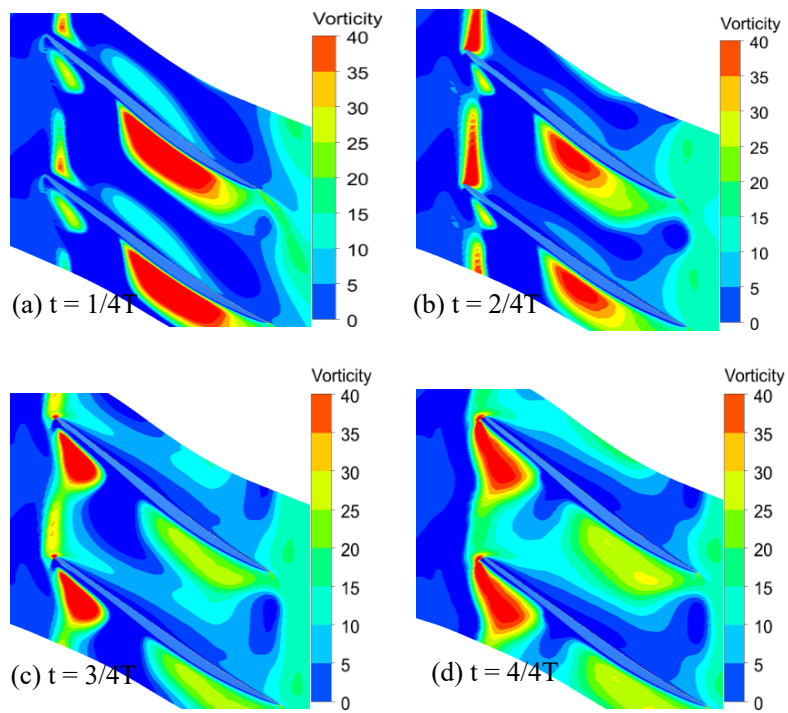


Fig. 9: Distribution of absolute vorticity at 98% span for 100% rotational speed close to stall condition.

Absolute vorticity distribution at 98% span for 100% rotational speed near to stall limit is shown in Fig. 9. The unsteady behavior at different time oscillation periods T of the absolute vorticity can be observed in Fig. 9. The coefficient of the absolute vorticity is explained as follows:

$$(8) \quad \xi_n = \frac{|\vec{\xi}|}{2\omega},$$

where ξ is vorticity vector and ω is angular velocity. At 100% rotational speed, the abrupt development of the TLV arose near the center of the blade chord due to downstream shock.

At the center of the vortex, recirculation ascended and vortex breakdown occurred. With the sudden expansion of the vortex breakdown, a large-scale fluctuation and reverse flow zone appeared. Absolute vorticity distribution at 98% span for 80% rotational speed close to stall limit is shown in Fig. 10. The unsteady actions at various time oscillation period T of the absolute vorticity can be observed in Fig. 10.

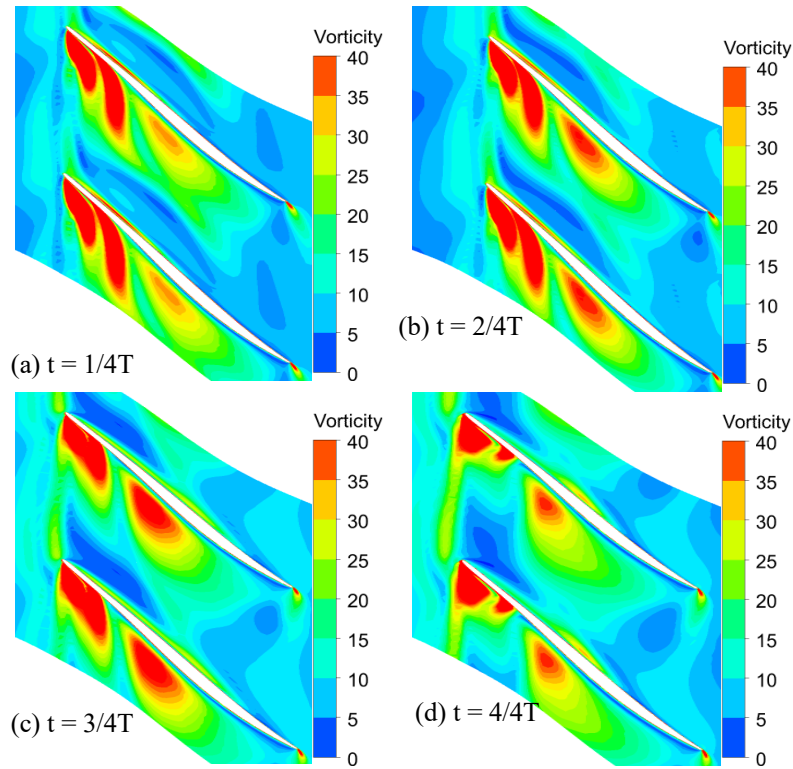


Fig. 10: Distribution of absolute vorticity at 98% span for 80% rotational speed close to stall condition.

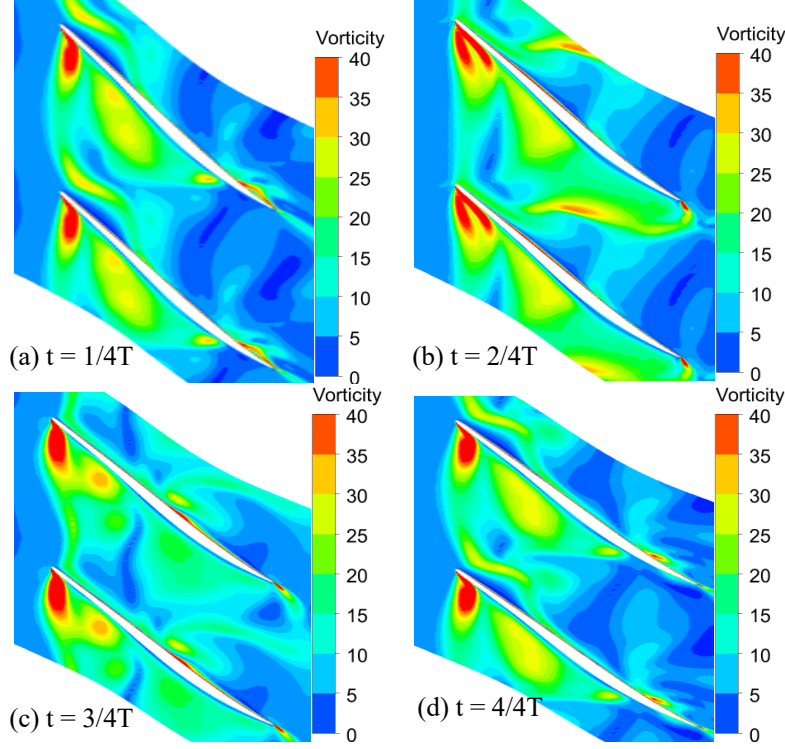


Fig. 11: Distribution of absolute vorticity at 98% span for 60% rotational speed close to stall condition.

At 80% rotational speed, the expansion of the TLV was larger than during the 100% rotational speed. The absolute vorticity's intense distribution was not present in the center of the TLV but appeared in different regions.

Absolute vorticity distribution at 98% span for 60% rotational speed near to stall condition is shown in Fig. 11 with different time oscillation period T . At 60% rotational speed, the absolute vorticity showed different behavior than the other two rotational speeds. The sudden development of the TLV arose in the tip leakage almost at 20% of the blade chord. The vortex breakdown occurred close to the leading edge of the compressor blade.

The cross flow slices of the normalized helicity for the three rotational speeds near stall conditions are depicted in Fig. 12. The streamlines of the tip leakage are also shown in Fig. 12. Normalized helicity H_n is explained as follows:

$$(9) \quad H_n = \frac{\bar{\xi} \bar{\omega}}{|\bar{\xi}| |\bar{\omega}|},$$

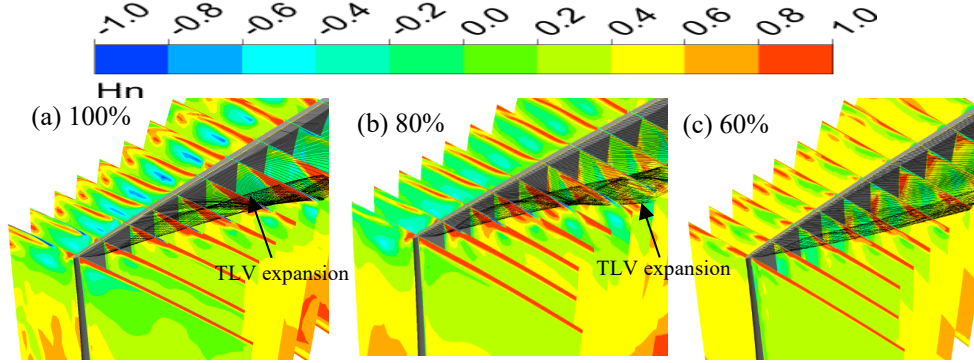


Fig. 12: Cross flow slices of normalized helicity and tip leakage flow at the blade tip for three rotational speeds near stall conditions.

where $\bar{\xi}$ and $\bar{\omega}$ indicate the absolute vorticity vector and relative flow velocity vector respectively. Normalized helicity H_n takes the magnitude of one in the streamwise vortex and quantitatively evaluates the vortex nature if the vorticity declines. At 100% rotational speed, the value of normalized helicity H_n is equal to one where the core of the TLV is located. The value of normalized helicity is equal to minus one for 80% rotational speed in the TLV expansion region. Some reverse flow occurred in the expansion region due to TLV and the same was reported by [7]. It can be concluded that significant variations occurred in the nature of the TLV due to large expansion. The value of normalized helicity H_n is equal to one at 60% rotational speed in the core of the TLV location and this showed resemblance with 100% rotational speed.

5 DESIGN GUIDELINES FOR AN EFFICIENT CASING TREATMENT

The simulations are carried out for 100, 80 and 60 percent illustrate that the stall mechanism change with the rotational speed. An efficient casing treatment for NASA rotor 37 has to enhance the local mass flow rate close to the blade tip upstream to minimize the aerodynamic loading for all rotational speeds. This significant fact prescribes the casing treatment application that can transport fluid upstream of the blade tip. It can be concluded that particular types of grooves should be designed to minimize aerodynamic loading. With different type of rotational speeds, the NASA rotor 37 casing treatment should influence the blade main flow as follow.

100% SPEED (DESIGN SPEED)

The numerical simulations for the smooth wall casing at 100% rotational speed elucidate that the dramatic increase of the vortex stagnation zone and blockage close to the upstream section of the pressure side of the blade is primarily responsible for

stability flow loss. Near the stall limit, the vortex stagnation zone extends around thirty to forty percent of the blade chord. The vortex stagnation happened due to the incoming flow steep angle of attack and TCF blockage close to the blade tip. Therefore, it is required to design an efficient casing treatment that can dampen the vortex stagnation zone effectively at 100% rotational speed. Generally, the blade tip leakage flow can only be controlled by manipulating the pressure gradient over the blade tip or adjacent to the front shock. To increase the flow stability of NASA rotor 37 at 100% rotational speed, it would be suitable to treat thirty to forty percent upstream of the blade chord because the mechanism of limiting stability flow takes place in this zone.

80% SPEED

The numerical simulations for the smooth wall casing at 80% rotational speed elucidate that the dramatic increase of vortex stagnation zone and blockage close to the upstream section of the pressure surface is primarily responsible for stability flow loss. The vortex stagnation zone and blockage take place in almost twenty percent of the blade chord. Close to stall limit, the vortex stagnation extent to adjacent blade suction surface. The shock is further pushed out to the LE of the blade due to blockage and VSZ. The angle of attack is steeper than the 100% rotational speed and the rotating stall occurs due to the incoming flow steep angle and TCF blockage. So, it is required to design efficient casing treatment at 80% rotational speed. Normally the blade TLF can only be controlled by manipulating the pressure gradient over the blade tip or adjacent to the front of shock. To increase the flow stability of NASA rotor 37 at 80% rotational speed, it would be appropriate to treat twenty to twenty-five percent upstream of the blade chord because the mechanism of limiting stability flow happened in this zone.

60% SPEED

The simulations for the smooth wall casing at 60% rotational speed show that the factor which limits the flow stability close to the blade tip at the stall is the extremely steep angle of attack of the incoming flow. The condition of the incoming flow caused aerodynamic overloading at the blade tip and produce a large separation zone at the suction side of the blade. The fluid of low energy leakage from the blade tip gap filled up the large separation zone and the incoming flow is continuously blocked till the complete rotating stall. The flow stability of the compressor at 60% rotational speed can only be increased if the casing treatment increases the mass flow rate close to the blade tip and hence turn down the angle of attack. A good technique is to increase the mass flow rate close to the end wall and further suppress the boundary layer separation at the trailing edge. To enhance the flow stability of NASA rotor 37

at 60% rotational speed, the axial groove should be fixed from ten to ninety percent of the blade chord to eliminate the low fluid from the blade passage in the downstream and reinject it at the upstream of the blade. The casing treatment should not decrease the efficiency at 60% rotational speed. This means that the quantity of leakage flow that passes over casing treatment should be small.

6 CONCLUSIONS

In the current research work Three passage, steady RANS, and URANS simulations were carried out on NASA rotor 37 to investigate the stall mechanism of the axial transonic compressor with three rotational speeds. The following valuable and significant conclusions are drawn from this research work.

- At 100% rotational speed, the interaction between the shockwave and TLV produced blockage which was the main cause of the rotating stall.
- At 80% rotational speed, the TLV breakdown that occurred downstream due to interaction with the shockwave caused the rotating stall.
- At 100% and 80% rotational speeds, a large blockage region was produced in the rotor tip due to the interaction between the shockwave and TLV.
- At 60% rotational speed, the TLV low energy fluid produced blockage in the compressor rotor tip which caused a rotating stall in the compressor. A large blockage boundary layer separation to the suction surface of the blade TE was some additional cause for the rotating stall.
- The tip leakage vortex breakdown occurred at 80% rotational speed near stall conditions at the blade tip of the rotor due to a large blockage expansion, while for 100% and 60% rotational speeds no such phenomena were observed.
- Mach number and axial velocity contours for 60% and 80% rotational speeds demonstrated that near stall conditions, the spilled flow appeared at the leading edge in the compressor rotor blade tip. The spilled flow appearing at the leading edge of the rotor increased when the rotational speed was decreased.
- The phenomena of spilled flow at the leading edge of the compressor rotor at 100% rotational speed near stall conditions have not appeared. The spilled phenomena did not increase at a higher speed.

REFERENCES

- [1] M. INOUE, M. KUROUMARU, M. FUKUHARA (1986) Behavior of tip leakage flow behind an axial compressor rotor. *Journal of Engineering for Gas Turbines and Power* **108**(1) 7-14.
- [2] I. DAY, C. FREEMAN (1994) The unstable behavior of low and high-speed compressors. *Journal of Turbomachinery* **116**(2) 194-201.
- [3] S. SCHLECHTRIEM, M. LÖTZERICH (1997) Breakdown of tip leakage vortices in compressors at flow conditions close to stall. Presented at: *the ASME International Gas Turbine and Aeroengine Congress and Exhibition*.
- [4] R.V. CHIMA (1998) Calculation of tip clearance effects in a transonic compressor rotor. *Journal of Turbomachinery* **120**(1) 131-140.
- [5] I. DAY, T. BREUER, J. ESCURET, M. CHERRETT, A. WILSON (1999) Stall inception and the prospects for active control in four high-speed compressor. *Journal of Turbomachinery* **121**(1) 18-27.
- [6] J. BALLMANN, W. HOFMANN (2012) Tip clearance vortex development and shock-vortex-interaction in a transonic axial compressor rotor. In: *40th AIAA Aerospace Sciences Meeting & Exhibition*.
- [7] K. YAMADA, K. FUNAZAKI, H. SASAKI (2008) Numerical investigation of relation between unsteady behavior of tip leakage vortex and rotating disturbance in a transonic axial compressor rotor. In: *Turbo Expo: Power for Land, Sea, and Air*.
- [8] R. BIOLLO, E. BENINI (2009) Shock/boundary-layer/tip-clearance interaction in a transonic rotor blade. *Journal of Propulsion and Power* **25**(3) 668-677.
- [9] Y. ZHANG, X. LU, W. CHU, J. ZHU (2010) Numerical investigation of the unsteady tip leakage flow and rotating stall inception in a transonic compressor. *Journal of Thermal Science* **19**(4) 310-317.
- [10] Q. LI, T. PAN, T. SUN, Z. LI, Y. GONG (2015) Experimental investigations on instability evolution in a transonic compressor at different rotor speeds. *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science* **229**(18) 3378-3391.
- [11] M. ZHANG, A. HOU (2017) Investigation on stall inception of axial compressor under inlet rotating distortion. *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science* **231**(10) 1859-1870.
- [12] Y. WU, G. AN, Z. CHEN, B. WANG (2019) Computational analysis of vortices near casing in a transonic axial compressor rotor. *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering* **233**(2) 710-724.
- [13] J. LANG, W. CHU, H. ZHANG, S. MA, X. CHEN (2017) Investigation of in-stall behavior in a transonic compressor rotor. In: *Turbo Expo: Power for Land, Sea, and Air*.
- [14] S. YE, Q. ZHAO, X. ZHOU, G. XI, J. XU (2019) The impact of circumferential casing grooves on rotating instability in a transonic axial compressor. *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering* **233**(8) 2868-2893.

- [15] A. LIU, Y. JU, C. ZHANG (2018) *Parallel simulation of aerodynamic instabilities in transonic axial compressor rotor. Journal of Propulsion and Power* **34**(6) 1561-1573.
- [16] M. HEWKIN-SMITH, G. PULLAN, S. GRIMSHAW, E. GREITZER, Z. SPAKOVSKY (2017) The role of tip leakage flow in spike-type rotating stall inception. In: *Turbo Expo: Power for Land, Sea, and Air*.
- [17] C. WEIXIONG, Y. WANG, H. WANG (2019) Investigation of the Unsteady Disturbance in Tip Region of a Contra-Rotating Compressor near Stall. *Journal of Thermal Science* **28**(5) 962-974.
- [18] J. DUNHAM (1998) CFD validation for propulsion system components, AGARD advisory report no. 355, advisory group on aerospace research and development, North Atlantic treaty organization. ISBN 92-836-1075-X.
- [19] NUMECA International (2009) IGG v8.9-1 and Autogrid5 v8.9-1, User Manual, NUMECA International, Brussels.
- [20] ANSYS 17 (2016) ANSYS CFX-solver theory guide, Release, ANSYS Inc.
- [21] A. NASEEM, J. BIN, Z. QUN, S.A. AHMAD, H. FAWZY (2019) Performance Enhancement of a Transonic Axial Flow Compressor with Circumferential Casing Grooves to Improve the Stall Margin. *Journal of Applied Fluid Mechanics* **13**(1) 221-32.
- [22] H. CHEN, X. HUANG, S. FU (2006) CFD investigation on stall mechanisms and casing treatment of a transonic compressor. In: *42nd AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit*.
- [23] K.L. SUDER (1996) Experimental investigation of the flow field in a transonic, axial flow compressor with respect to the development of blockage and loss. PhD thesis.