Amirkabir Journal of Mechanical Engineering

Amirkabir J. Mech. Eng., 51(2) (2019) 137-139
DOI: 10.22060/mej.2018.13976.5768

Numerical Simulation of Tip Leakage Flow Structure in the Transonic Axial Compressor in Different Performance Conditions

S. Abbasi*, E. Behroozizade

Department of Mechanical Engineering, Arak University of Technology, Arak, Iran

ABSTRACT: In this paper tip leakage flow structure of a transonic axial compressor rotor in different performance conditions will be simulated. Results have been presented according to a 3D numerical simulation of the viscous flow and solving Navier-Stokes, Continuity and energy equations using Ansys-CFX software. Initially, performance curves have been derived and compared with experimental results and have shown good agreement. Then, results have been obtained from three mass flow rates including design, choke and near stall conditions. The results have indicated that reduction of mass flow rate from chock to stall condition leads to increase in the tip leakage flow strength. This phenomena causes to more loss, especially in the blade tip region. In addition, position of shock line moves to upstream when the mass flow rate decreases. The tip leakage flow, shock and main flow contact cause a complicated flow structure near stall condition which leads to increase in entropy, vortex flow and blockage.

Review History:

Received: 21 January 2018
Revised: 25 May 2018
Accepted: 24 June 2018
Available Online: 6 July 2018

Keywords:

Axial compressor
Numerical simulation
Performance curve
Performance conditions

1- Introduction

Flow in axial compressors is 3*D* and its complexity increases due to various flow interference. In the subsonic compressors, a 3*D* complex flow is created as a result of main flow, blade boundary layer and blade rotation. In transonic compressors, flow complexity increases due to interaction of shock waves with various flow. Tip leakage flows, stall and surge phenomena cause to limit an axial compressor performance. Therefore, studying flow in axial compressors is of great importance. Many numerical and experimental researches has been conducted to investigate the effects of different conditions on axial compressors performance and instability.

Inoue et al. [1] studied two different tip clearance size in an axial compressor. Results showed that by increasing the tip clearance length, the vortex flows increases due to interaction of tip leakage flow and main inflow. Liu et al. [2] performed a study on a low speed axial compressor rotor. They observed formation and dissipation of vortex flow in vicinity of stall condition.

The literature survey indicates that most of the researches have focused on the low speed rotors. No study has been conducted on the complexity of shock waves, shock interference and tip leakage flow .While most compressors work in transonic conditions. Therefore, investigation of compressor performance conditions have created gaps for further researches. It is necessary to investigate performance of an axial compressor and complex flow structure in different conditions (from chock to stall) to consider capable method to enhance performance and tip leakage flow. Therefore, flow analysis in a transonic axial compressor rotor has been studied in this paper. Also, details of flow structure in different performance conditions will be studied and compared. The interaction of boundary layer, shock wave and tip leakage flow study in different conditions and its effects on some parameters such as pressure, Mach number and entropy cause a better comprehension of various phenomena occurred in transonic axial compressor.

Corresponding author, E-mail: s_abbasi@arakut.ac.ir

2- Numerical Simulation

2- 1- Calculation geometry

NASA Rotor 37 geometry [4] is used for simulation in this paper. NASA Rotor 37, as a transonic axial compressor, has been studied by many researchers [3-5].

Numerical simulation has been carried out just for one compressor passage. A passage contains of three parts including rotor inlet, rotor blade and rotor outlet.

2- 2- Meshing

A structured mesh has been adopted for the passage. A passage includes 100 streamwise nodes, 30 spanwise nodes and 69 pitchwise nodes. The radial space between the blade tip and compressor casing has been divided into 12 nodes. Meshing near walls has been carried out in a sense that $y^+<5$ to evaluate the flux viscous amount using no slip and adiabatic wall condition. In the present study, a passage consists of 290514 elements.

2- 3- Solving preferences

In the present study the commercial software (Ansys-CFX) has been employed. This flow solver is a three-dimensional, viscous. The computation method utilizes

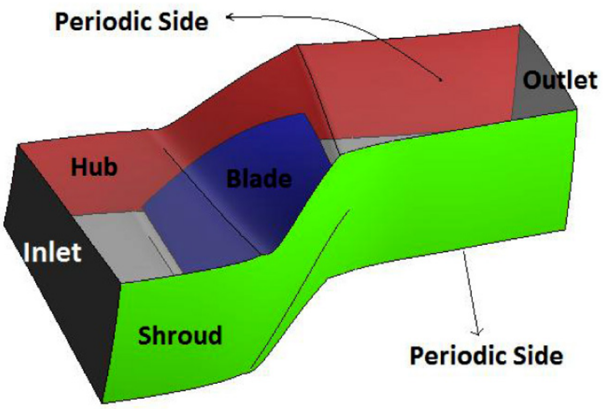

Fig. 1. Geometry and boundary conditions of NASA Rotor 37

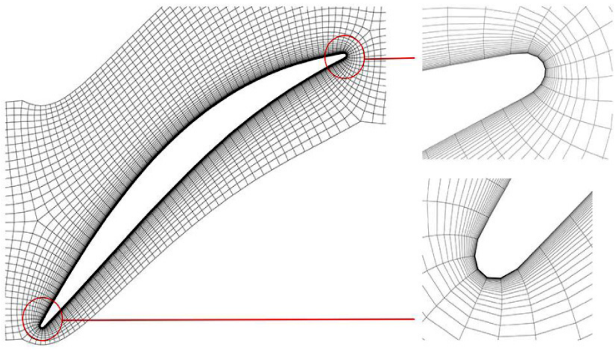

Fig. 2. Rotor blade meshing near hub region

a finite volume scheme to solve Momentum, Continuity and Energy equations. In order to find Reynolds tensors and eddy viscosity, the *k*-*ω*-SST turbulence model has been used. Simulation is in steady state and the air fluid has been considered as ideal gas. Rotating coordinate system is used in the present study. In the rotor inlet boundary, Total pressure and total temperature are set to 100 kPa and 15ºC, respectively. In the rotor outlet boundary, static pressure is used. For the repeated boundaries, the periodic boundary conditions have been imposed. Different outlet static pressure has been set to investigate its effects on different performance conditions. Outlet static pressure increases from choke to stall condition.

3- Results and Discussion

3- 1- Performance curves and results validation

Compressor performance curve (total pressure ratio vs mass flow rate) has been presented in Fig. 3. The experimental results of the transonic axial compressor have been added to Fig 3 [5]. There is a good agreement between experimental results with that of the numerical ones.

Table 1 has been provided to provide a better comparison of the results. Maximum error of the present results is 3.75% that is suitable.

3- 2- Results and discussion

Relative pressure contour and flow streamlines in different conditions have been presented in Fig. 4. The mass flow rate reduction is accompanied by pressure ratio increase that causes stronger tip leakage flow. By increasing tip leakage flow and decreasing the main flow, flows move to upstream and these phenomena can be seen near the rotor leading edge.

In Fig. 5, the entropy distribution in the streamwise direction is shown at different condition. The flow entropy before arriving to the rotor blade is equal in all three mass flow rates. By flow passing, the flow contacts the blade and entropy increases rapidly. One can be recognized that in the near stall condition increase of entropy occurs sooner relative to design and chock conditions.

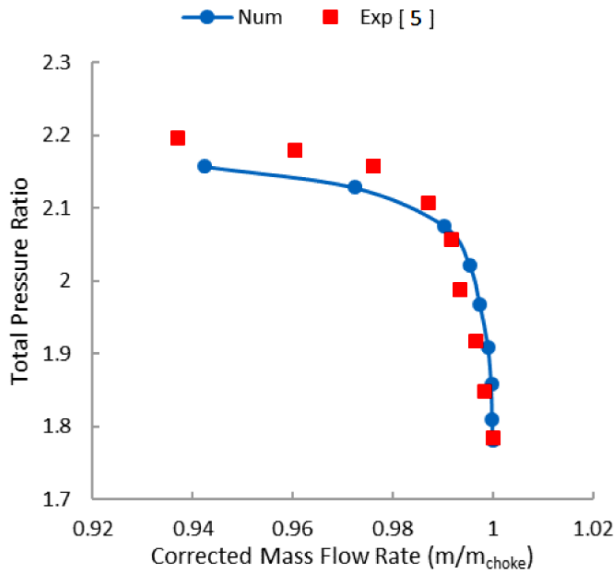

Fig. 3. Compressor performance total pressure ratio curve

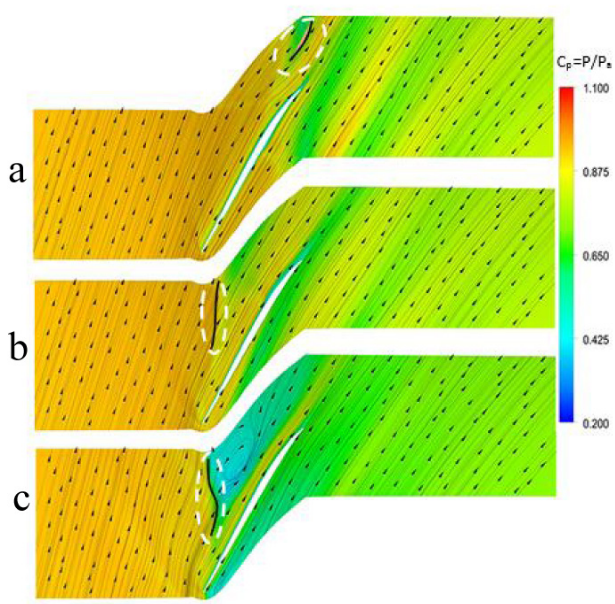

Fig. 4. Relative total pressure contour in span 0.97 with streamlines in three mass flow rates of (a) choke, (b) design and (c) near stall.

Table 1. Compare numerical and experimental results in different performance conditions

Performance Condition	Corrected Mass Flow Rate	Total Pressure Ratio		Error (%)	Adiabatic Efficiency		Error (%)
		Num	Exp [5]		Num	Exp [5]	
Choke	1	1.784	1.784	0	84.48	84.302	0.21
Design	0.9917	2.063	2.056	0.34	85.964	87.804	2.09
Near Stall	0.9604	2.13	2.213	3.75	84.836	86.498	1.92

4- Conclusion

Flow in a transonic axial compressor was simulated numerically in different performance conditions using Ansys-CFX software. The results validation determined maximum error about 3.75% which is desirable. The reverse flow was not seen in design condition. But in the choke condition, the reverse flow was observed near the rotor hub and in the near stall condition in the rotor tip region. The Relative total pressure contour indicated that the interface of main inflow and tip leakage flow moves to upstream when the mass flow rate decreases. Moreover, the relative Mach number distribution represented that the movement of shock wave line is accompanied with reducing the mass flow rate.

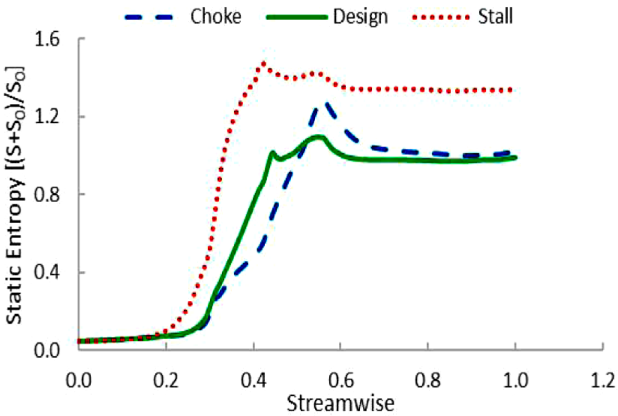

Fig. 5. Static entropy changes along the flow direction in different performance conditions

References

[1] M. Inoue, M. Kuroumaru, Structure of Tip Clearance Flow in an Isolated Axial Compressor Rotor, Journal of Turbomachinery, 111(3) (1989) 250-256.

[2] B. Liu, H. Wang, H. Liu, H. Yu, H. Jiang, M. Chen, Experimental Investigation of Unsteady Flow Field in the Tip Region of an Axial Compressor Rotor Passage at Near Stall Condition With SPIV, (36894) (2003) 877-888.

[3] C.-T. Dinh, M.-W. Heo, K.-Y. Kim, Aerodynamic performance of transonic axial compressor with a casing groove combined with blade tip injection and ejection, Aerospace Science and Technology, 46 (2015) 176-187.

[4] L. Reid, R. D. Moore, Design and overall performance of four highly loaded, high speed inlet stages for an advanced high-pressure-ratio core compressor, 1978.

[5] R. D. Moore, L. Reid, Performance of single-stage axial-flow transonic compressor with rotor and stator aspect ratios of 1.19 and 1.26 respectively, and with design pressure ratio of 2.05, 1980.

Please cite this article using:

S. Abbasi, E. Behroozizade, Numerical Simulation of Tip Leakage Flow Structure in the Transonic Axial Compressor in Different Performance Conditions, *Amirkabir J. Mech. Eng.*, 51(2) (2019) 137-139.

DOI: 10.22060/mej.2018.13976.5768