CFD ANALYSIS OF COMPRESSOR BLADE STALLING AT 5 DEGREE ANGLE OF ATTACK

Meet Patel[1]

[1]*Department of Aeronautical Engineering, Sardar Vallabhbhai Patel Institute of Technology, Gujarat, India*

ABSTRACT : Stalling is one of the common phenomena observed when blades/airfoils are at higher angle of attack. Compressor blades also get stall due various parameters. NACA 65 series airfoils are used as compressor blades. Our project is based on the analysis of compressor blade stalling. In the previoussemester we have gathered the necessary data required for the analysis and also performed CFD analysis over NACA 65(1)-212 airfoil. We did the analysis but there was error in grid generation so we were not able to get the fruitful results. In this semester we have identified the errors and we have performed the analysis over the NACA 65(1)-412 airfoil. In this project the analysis is done by giving different angles to the inlet flow direction. We have performed analysis at angles 5 degree angle of attacks. The stalling phenomena is observed by observing the velocity contours.

INTRODUCTION

In this chapter the basics of our project is discussed like, stall and its types, surge, effects of stall, aim and role of our project.

What is stall?

Stall is a phenomena which is caused by the local disruption in airflow inside compressor of jet engine of aircrafts. The complete disruption of airflow caused due to stall inside compressor is called surge. Types of stall: Axial compressors suffers from two possible modes of unstable operation

1. Rotating Stall: Non-axisymmetric and aperiodic

2. Surge: Axisymmetric and periodic

Rotating Stall: Progression around the blade annulus of a stall pattern, in which one or more adjacent blade passages are instantaneously stalled, then are cleared for unstalled flow as the stall cell progresses is called rotating stall. Rotating stall causes alternate loading and unloading of the blade resulting in fatigue failure.

SURGE: Surge line is a locus of all the surge points of multistage compressor. Surge basically involves fluctuation of fluid back and forth in annulus of compressor. It is the axial fluctuations of the fluid and entire annulus of compressor is affected that is why axisymmetric. Surge line denotes locus of unstable operation of compressor. Surge is characterized by violent, periodic oscillations in flow. Surge might lead to flame blow out in combustion chamber. Surge can lead to substantial damage to compressors and must be avoided. The operating line of the compressor is therefore kept slightly away from the surge line: surge margin.

PARAMETERS CAUSING STALL

There are various parameters which causes the compressor stalling.

• Imbalance between the airflow supply and airflow demand in other words, a pressure ratio that is incompatible with the engine RPM. Compressor stall cause the air flowing through the compressor to slow down or stagnate and sometimes result in reverse flow.

• Foreign Object Damage(FOD) including bird strike.

• Worn, dirty or contaminated compressor components.

• In-Flight icing.

• A/c operation outside of engine design envelope including extreme flight maneuvers. • Improper engine handling. • Flow separation due to high angle of incidence because of non-uniform flow.

Stalling can be self correcting or steady. It causes reduction in engine thrust and is likely to produce "backfire" like sound due to reverse airflow. Sometimes flame may come out from either or both from inlet and exaust. Compressor stall also occurs due to the tip leakage flow from the tip of the compressor rotor blades which is unavoidable. It is more at rotor tip and beomes negligible after 80% span of rotor blade from tip. The parameter on which we are focusing in this project is the flow separation due to high angle of incidence.

ANALYSIS OF COMPRESSOR BLADE

The airfoil used in our project is NACA 65(1)-412. We have downloaded the coordinates of NACA 65(1)-412 aifoil from airfoiltools.com and imported it to ANSYS fluent from excel sheet. Using concept and surfacing the complete airfoil surface was generated.

GEOMETRY :

We have imported the geometry using option concept- 3D curve from coordinated file. After surfacing the airfoil geometry we have drawn the upper and lower wall boundaries by taking the mean camber line to create the 2-D cascade at a distance of 0.9c above and below the blade geometry. To generate the flow domain we have taken a distance of 1.5c in front of the leading edge and 2c at the back of the trailing edge. It gives the domain a look like a compressor casacade. The upper wall boundary behaves like the lower surface of the forward blade and similarly the lower wall boundary behaves like the upper surfaceof the aft blade. Airfoil chord length is given as 1m. To create a surface 'surface from edges' tool is selected and then all the edges including domain and blade are selected. By using the Boolean function the body of the blade is subtracted from the domain to avoid getting grid over the blade cross section. Create a rectangle that starts at a distance of 0.35m in front of the leading edge till the trailing edge. Height above and below the axis should be kept 0.35m. Using fillet option give radius of 0.3m in front of the leading edge that passes through domain. Create a vertical line at some distance from the leading edge that passes through the domain. Create another line at trailing edge perpendicular to x-axis passing through domain. Draw lines horizontal from leading edge and trailing edge passing through the domain. Now we use the 'face split' option from tools to select surface body as target surface and select all the lines created with the geometry created in front of airfoil to divide the domain into number of faces. Our geometry is prepared.

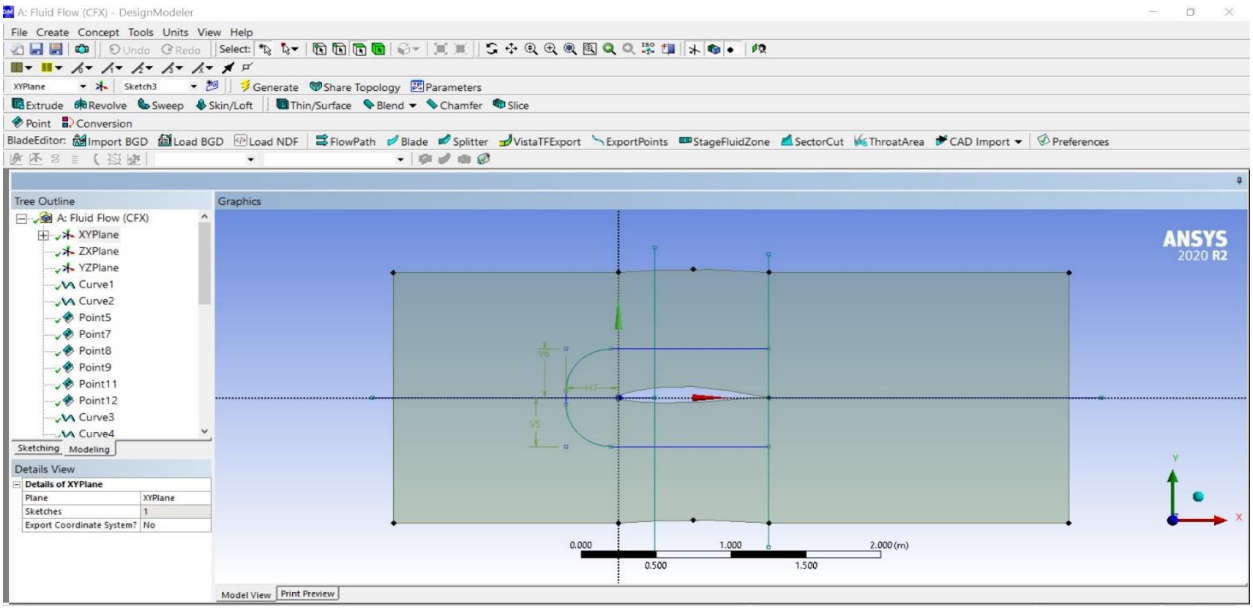
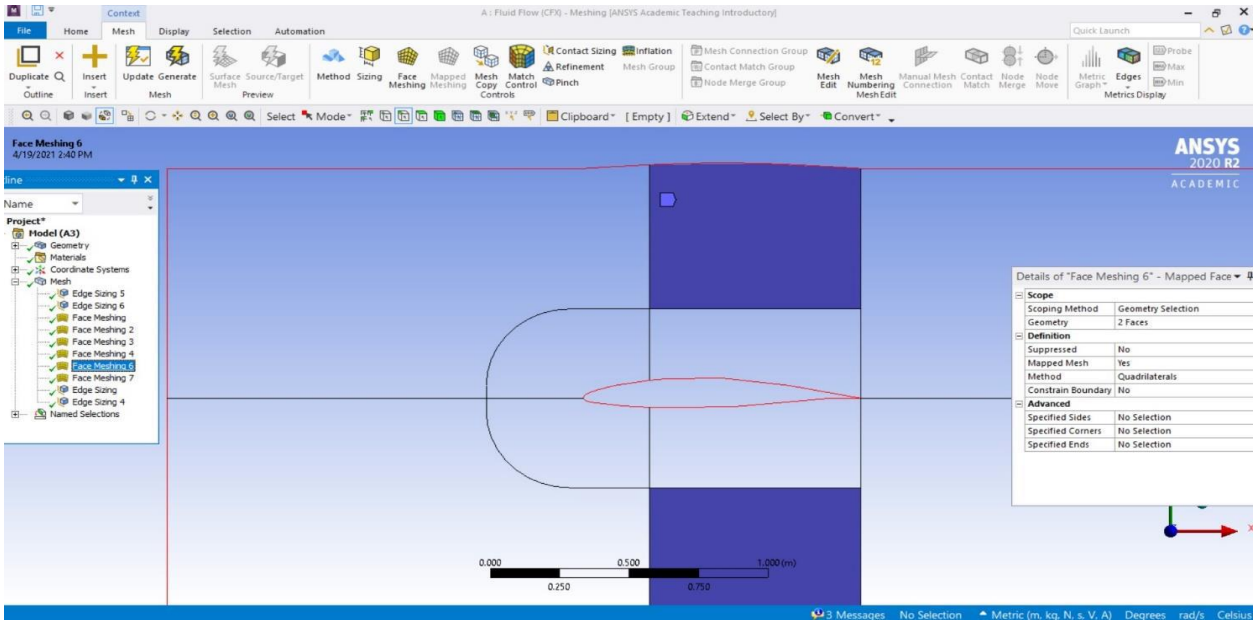


Fig: Geometry (ANSYS Workbench)

MESHING:

In meshing create a default mesh to understand how to proceed. Apply sizing from mesh menu. (a) Select the edges of the part of the airfoil separated by vertical line which is at some distance from the leading edge. Set the number of elements as 0.02 and behaviour as 'Hard'. (b) Similarly apply edge sizing to the lines above upper side of the blade within geometry created surrounding the blade. Set number of divisions as 20 and biasing such that cells are close to blade and far as we move away from the surface. Set bias factor as 28. Apply face meshing to each and every face. Below figure is of face meshing.



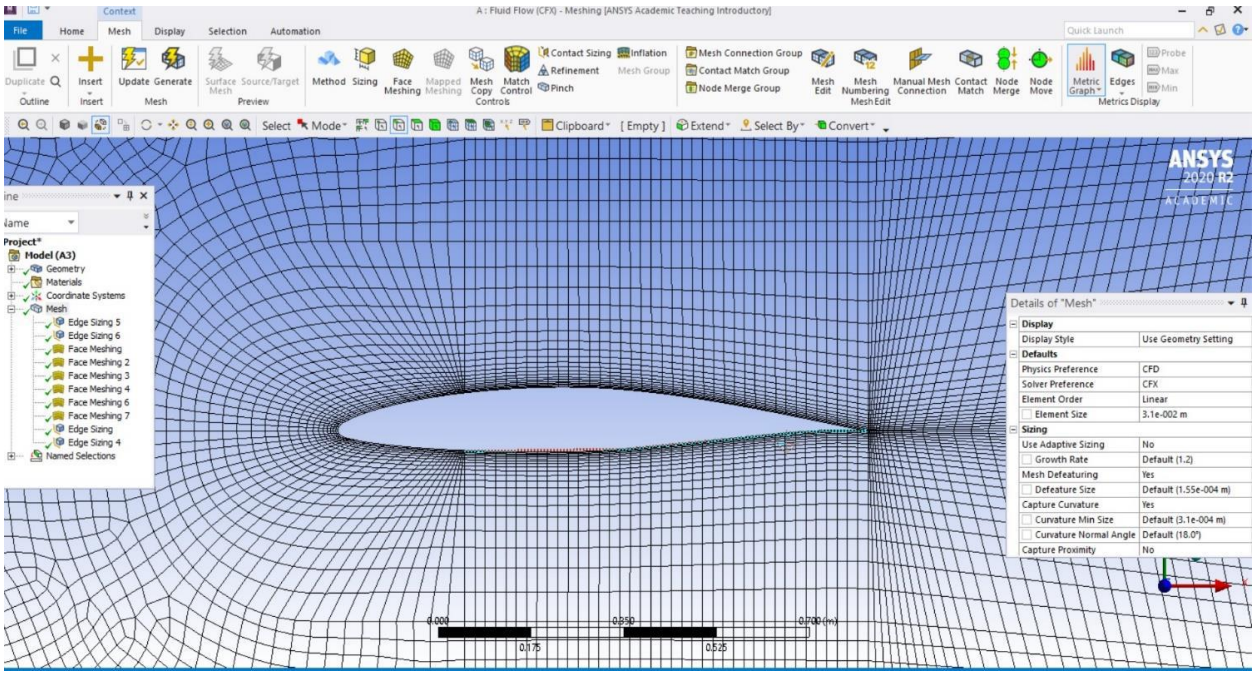


Fig: Meshing 2 (ANSYS Workbench)

SETUP:

Now we go to setup after generating the mesh. Energy equation has to be turned on as there will be change in temperature at outlet. Select K-OMEGA as the modal to analyse the flow over blade. In materials we choose air. Instead of constant density we choose ideal gas as the flow will be compressible inside the compressor as there are considerable changes in pressure and temperature inside compressor.

BOUNDARY CONDITIONS

: Boundary conditions at the inlet was given as velocity = 33.33m/s and initial temperature as 300K. At inlet gauge pressure is zero as atmospheric pressure at inlet is 101325Pa. At outlet we specify gauge pressure as 303Pa as Atmospheric pressure at outlet is 101628Pa. Pressure Ratio: 1.0033. INTENSITY AND LENGTH SCALE method is used. At the walls no slip boundary conditions are given. Method used for the CFD analysis is COUPLED SCHEME with SECOND ORDER UPWIND SCHEME in all parameters. Report Definition – New – Force Variable – Cl add Similarly add Cd as we want graph of Cl vs Cd.

INITIALIZATION:

Hybrid initialization method is used for initialization. Initial values are given as, Pressure: 101325 Pa X velocity: 33.33m/s Y velocity: 0m/s Number of iterations given for running the calculations are 400.

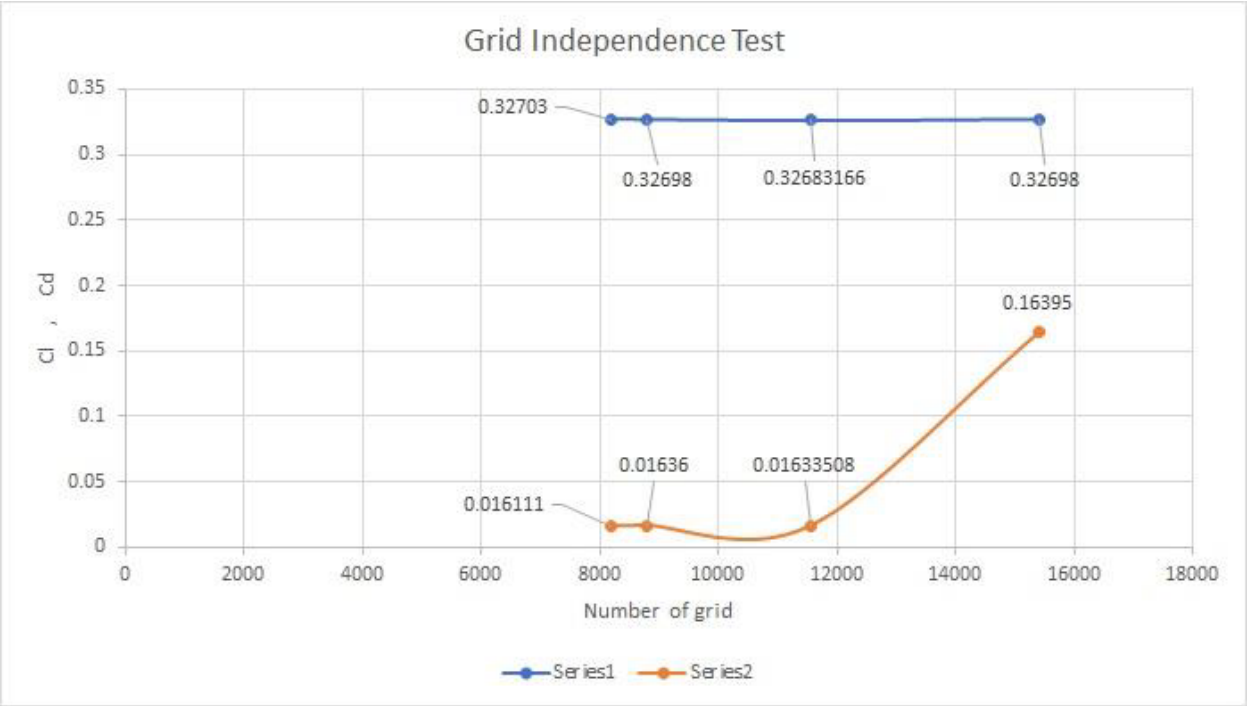


Fig. Grid Independence Test Graph

5 DEGREE ANGLE OF ATTACK

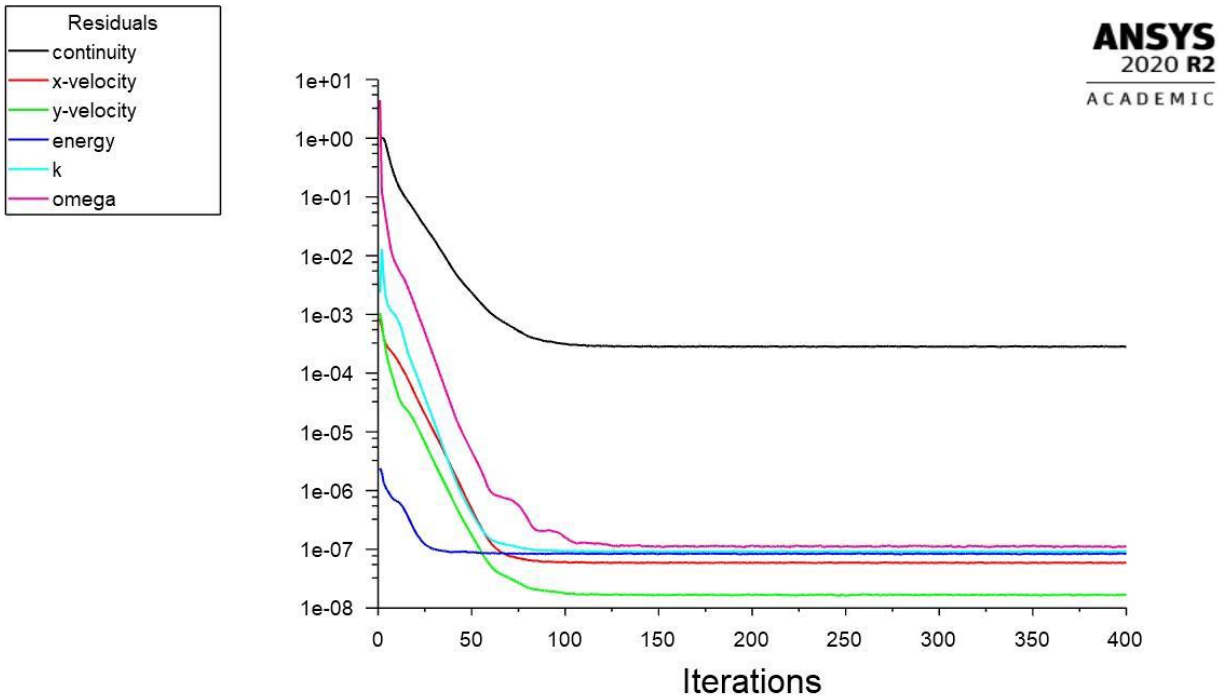


Fig: Run Calculation (ANSYS Workbench)

Fig: Velocity Contour (ANSYS Workbench)

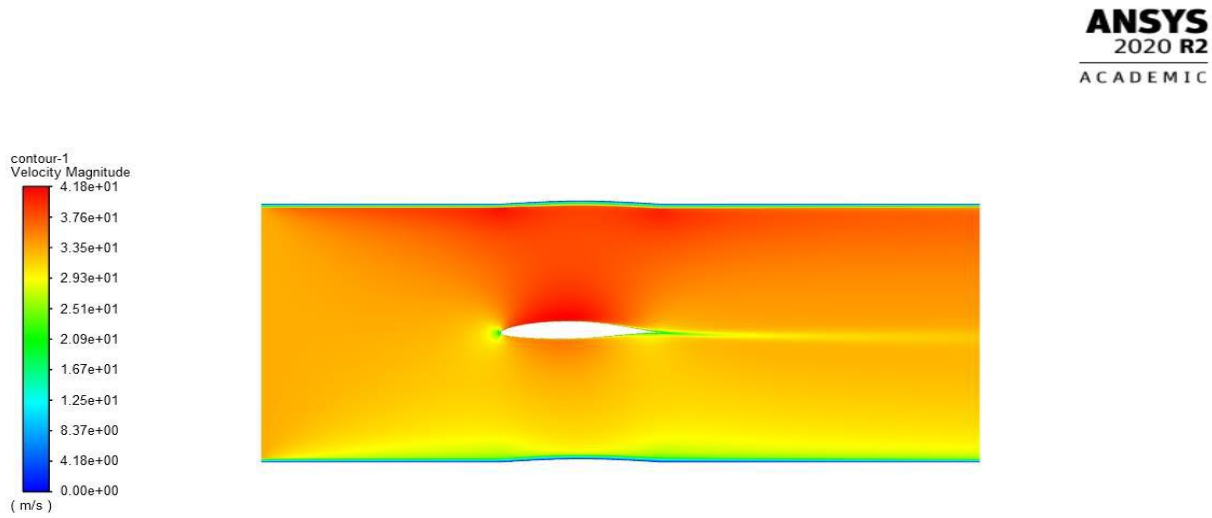
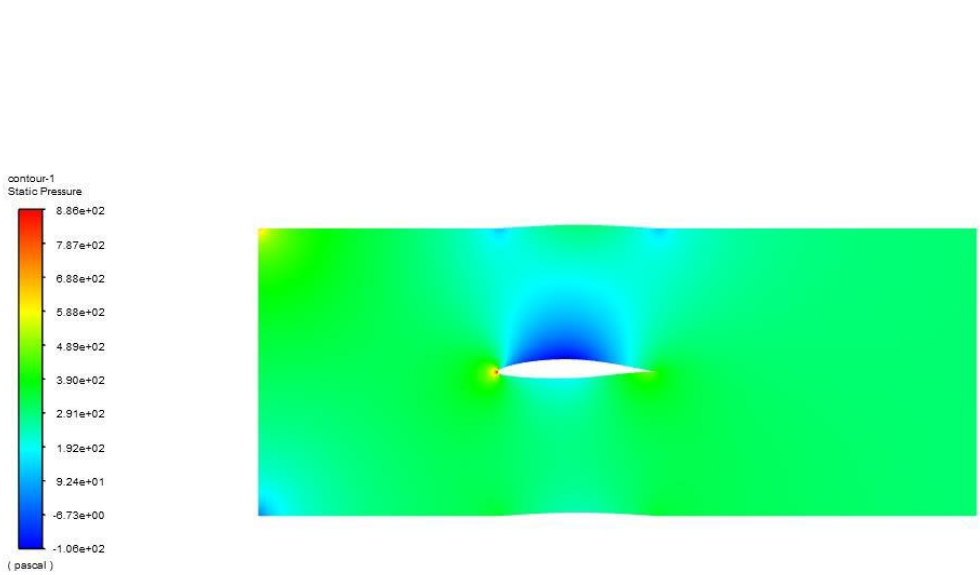


Fig: Static Pressure Contour (ANSYS Workbench)



CONCLUSIONS

• We have analyzed the flow over NACA 65(1)-412 airfoil using Cascade Geometry with the help of ANSYS Fluent at angles 5, 12, 20, 28, 39 degrees.

• By analyzing and observing the flow over the blade in a linear cascade at different angles of attack, we conclude that flow separation occurs at higher angle of attack which leads to stalling of the compressor blade.

• Also at higher angle of attack, static pressure about the blade drops drastically which causes loss in pressure ratio leading to surging of the compressor.

• To avoid such situations, the incoming flow must be incident at low angles of attack. For the same, proper selection and installation of Inlet Guide Vanes must be done, regular maintenance and checks should be carried out, off design operation should be avoided.

REFERENCES

[1] www.airfoiltools.com

[2] Compressor Blade Design – Jerry M. Seitzman Professor

[3] Aerodynamic analysis of NACA 65-2012 airfoils at different attack angles with Computational Fluid Dynamics(CFD) – Kiumars Khani Aminjan.

[4] The blade compressor geometry, performance and applications – C Hudson.

[5] Axial compressor stall and surge prediction by measurements – H. Honen.

[6] Observations of propagating stall in axial floe compressor – T. Laura and W.D. Rannie

[7] Computational study of Rotating stall inception in Axial-Compressor – L.He

[8] CFD analysis to understand the flow behavior of a single stage transonic axial flow compressor – M T Shobhavathy and Premaraka Hanoca.

[9] Cascade viscous flow analysis using the Navier-Stokes equations – Roger L. Davis, Ron-Ho Ni and James E. Carter.

[10] Inviscid and viscous solution for airfoil/cascade flows using a locally implicit algorithm or adaptive meshes – C.J. Hwang and J,L,Liu.

[11] Numerical simulation of rotating stall in a subsonic compressor – N. Gourdian, S. Burguburu, F. Leboeuf, H. Miton.

[12] Propogation of stall in the compressor blade row – Frank E. Marble.

[13] A computational model for rotating stall and inlet distortion in multistage compressor – Yifang Gong.

[14] Computation of tip leakage flow in a linear compressor cascade with a second moment turbulence closure – Domenico Borello, Kemal Hanjalic, Franco Rispoli.