Two dimensional rotor blade numerical analysis using ANSYS

Shlok Anand, Satvik Shenoy and Srinivas G*

Department of Aeronautical and Automobile Engineering, Manipal Institute of Technology, Manipal Academy of Higher Education (MAHE), India

*srinivasle06@gmail.com

Abstract. Turbo-pump is an essential component in any rocket engine. The airfoil (NGTE 10C4 series) section of a rotor blade in the SSME (Space Shuttle Main Engine) was modelled along with a far-field of suitable shape and dimensions. The model was suitably discretized with proper amount of refinement, etc. A numerical simulation was then carried-out using ANSYS software with several iterations of varying inlet boundary conditions to check for any improvement in the performance of the rotor blade. Inlet velocity for optimum functionality of the blade was identified.

Keywords: Turbo-pumps, rotor, pressure-ratio.

1. INTRODUCTION

1.1. Introduction to non-air-breathing engines

Air breathing engines are a class of propulsion systems which use a combusted mixture of air and fuel to propel the adjoined structure (mainly an aircraft). In contrast to this there is another set of engines which do not use air as one of its propellant mixture constituents. Such engines carry the propellants required and are referred to as non-air breathing engines. These propellants undergo combustion and are propelled through a convergent-divergent nozzle to produce thrust. Non-air breathing engines do not have an inlet. The fuel and oxidizer are provided to the combustion chamber by pipes and pumps. In the combustion chamber the propellants are burnt to produce the thrust. Optimizing a rocket engine can be done in various ways, one of the ways is to improve turbines and pumps that are used to supply the propellants from their respective tanks to the combustion chamber.

1.2. Importance of turbines:

A turbine helps to decrease the (static) pressure of engine gas. Total pressure remains unchanged thus increasing the velocity (dynamic pressure) of the gas. In the present era, turbines contribute to being an important part of aircraft as well as rocket propulsion system. Gas-turbines (jet-engine) and turbo-pumps (rocket engine) are two of the many applications of turbines. Turbo-pumps make use of several stages (rotor and stator blades in series) to feed liquid fuel and oxidizer to the thrust chamber of a rocket engine.
The Space Shuttle Main Engine (SSME) (see figure 1) uses an RS-25 (Rocketdyne engine) with cryogenic liquid hydrogen as fuel and liquid oxygen (LOx) as oxidizer. A rotor blade of the LOx turbo-pump in the SSME was studied through a 2D numerical simulation.

### 1.3. Different Types of Rocket Engine Turbines

Rocket engines can be classified on the basis of number of stages of the turbine. Each set of consecutive rows of rotor and stator blades in a turbine is called a stage. Turbines can be 3-stage, 5-stage, 7-stage or even 9-stage depending on the desired temperature and pressure ratio. Increasing the number of stages does not necessarily give higher efficiency. The temperature ratio of any stage in a turbine is same and constant. The pressure ratio is calculated by the isentropic relations. Although not linear, the pressure can be noted to be decreasing in every stage.

### 1.4. Recent Developments

In the present era Liquid Propellant Rocket (LPR), only a small portion of the fuel and oxidizer is passed through the pre-burner. The “full-flow staged cycle” is an important advancement. The entire fuel and oxidizer flows through the pre-burners but only a small amount is consumed. The increased mass flow rate drives the turbo-pump to give higher pressure ratios with slightly lower temperature. The performance of the turbo-pump is increased.

LE-9 is Japan’s most recent booster engine and is an important component of the H3 launch vehicle. The system is reliable, high performing and inexpensive all at the same time. The first test flight is scheduled to be in the year 2020.

Liquid Natural Gas (LNG) which is basically liquid oxygen and liquid methane is the propellant in the Liquid Upper Stage Demonstrator Engine (LUMEN). The working is based on the expander cycle and the German Aerospace Centre desires to provide a test bed for any future components.

### 2. Literature Review

D.G. Ainley et al. [1] and J. Dunham et al. [8] used a numerical approach to calculate the performance of a conventional axial flow turbine. Derived data from the study of several turbine tests in a separate research paper was used in this method. The method proved to be consistent with the data received by the National Gas Turbine Establishment (N.G.T.E.) and the efficiency was calculated to be within the vicinity of $\pm 2.0\%$ error. Improvements were then made in the numerical method.

James Boynton et al. [2] experimented to conclude on the effect of tip clearance on the functioning of an axial hydraulic turbine. The maximum clearance given was up to 8% of passage height and the trials were conducted in water. The shrouded blades provided the highest efficiency at design clearance and design operation. The efficiency of the second stage was about 10 points lower than the first one. The reduction in the efficiency with each percent increase in the tip clearance were approximately 1.2% for the first stage and 4.1% in the repeating stages. The experimental results were validated by a numerical analysis.

B. Lakshminarayana et al. [3] used both theoretical and experimental methods to examine the effect of a gap in an isolated airfoil spanning a tunnel. Flow models of each a small gap ratio and a considerably large gap ratio were recommended. Increase in drag and decrease in lift for small gap ratios were noted to be very small. Due to flow circulation along the gaps, there were turbulences at the tip. It was noted that with change in gap size, the location and strength of the tip vortices vary. At large chord-gap ratios, the chord is not aligned with the direction of the leakage flow. The theoretical
predictions match with the experimental observations made.

C. H. Sieverding et al. [4] experimented on secondary flows during the period, 1975-1985. The research article, although outdated, holds good relevance in present era studies. A detailed analysis about secondary flow vortex structures was carried out. The effects on end wall boundary layer characteristics as well as losses through straight turbine blade passages were studied. The article lacks in providing sufficient numerical or theoretical validation to the results obtained.

S. Sjolander et al. [5] have examined the configuration of the three-dimensional tip leakage flow and its effect on the blade loading in a large series (or cascade) of turbine blades. The variation in tip clearance was 0-2.86% of blade chord. Furthermore, to validate the data obtained, oil and smoke were used for an extensive visualization. Not much information was provided about the effect of the inlet boundary layer thickness, load distribution and the blade geometry. The reduction in the pressure induced by the leakage vortices on the suction side was larger than the pressure side.

A. K. Saha et al. [6] numerically studied the effect of a pressure side winglet on the flow and the heat transfer over a blade tip. A flat tip and a squealer tip were studied. A finite-difference RANS code and a 2-equation model for turbulence was used for calculation purposes. The pressure-side winglet for a flat-tip led to considerable decreases in the local heat transfer coefficient on the tip of the blade. The winglet showed some notable decrease in the strength of leakage flow and the associated vorticity. For the squealer tip, the winglet produced only minor improvements. For the suction sided squealer tip, the pressure side winglet provides no substantial improvement.

C. M. Rhie et al. [7] presented a finite volume numerical solution for the two-dimensional incompressible, steady Navier-Stokes equation, in general curvilinear coordinates. Turbulent flow models were applied over different airfoils at different orientations with and without edge separation. The flow process is described by the k-epsilon model. An ordinary grid system is used rather than a staggered grid. The numerical simulation carried an objective to study the flow over an isolated airfoil and to check whether the results concur with the experimental data. The airfoils used were NACA 0012 and NACA 4412 at angles of attack 0 and 6 degrees similar to [13,14]. Accuracy of the solution was observed to be highly dependent on degree of resolution of the strong leading and trailing edge gradients.

![Figure 2](image2.png)

**Figure 2**: Far-Field (with breadth of 6 times the chord length)

![Figure 3](image3.png)

**Figure 3**: Airfoil section of the rotor blade of LOX turbo-pump of the SSME
This detailed analysis of the method brought about certain changes to the formula for tip clearance loss predictions. In addition to this the accuracy of the improved results and formula were also checked. This method is a very simple approach but holds great value during preliminary design.

S. C. Kacker et al. [9] worked on mean line loss system (based on work by Ainley-Mathieson), a prediction method capable of estimating the design point efficiencies of axial turbines in present era gas turbine engines. The method was discovered against the ‘Smith’s chart’ and the known efficiencies of all 33 turbines of the then relevant designs. Regardless the fact that the logical tools available for the optimum design of turbine blading are very distinguished, the optimization of the turbine gas path and efficiency predictions will always require a mean line design.

![Meshing of the airfoil section](image)

**Figure 4.** Meshing of the airfoil section

3. Methodology

3.1 Modelling

Airfoil data [3] was extracted. The airfoil was modelled using ANSYS Space-Claim modelling software. (see figure 3) A suitable far-field was modelled around the airfoil as shown in the figure 2. Boolean operations were carried out to separate the tool body and target body, in order to optimize the area of interest.

3.2 Meshing

The two-dimensional model created in the previous section is a continuous surface and has infinite degrees of freedom which makes the solving of such type of surfaces a challenge. To solve, a mesh is laid on the said surface to reduce the degree of freedom to a finite number which makes the solution of such problems possible. Accordingly, the domain is broken down into various pieces. Each piece represents an element. The model was given a layout of meshes, the scope of which was the body around the airfoil(figure 4) up to the boundaries of the far field and the method used was ‘all triangles method’. After using the said method, edge sizing was employed at the edges of the airfoil to have a finer meshing near the key subject. The definition type was ‘element size’ (element size was set at 1.e-02). In addition to these, two sets of refinements have been given to the far field surface and the edges of the airfoil (the value of refinement was set to 2).

3.3 Boundary Conditions

Applied-

To define the boundary conditions, certain perquisites are required, such as identifying the location of the boundaries and supplying correct information about the boundaries. The data required at the boundary is dependent on the boundary type and the physical model applied. The data provided as the boundary condition must be precise or at least be reasonably approximated. The boundary conditions were derived from [10].

During laying of meshes in the two-dimensional design the naming of certain sections of the heterogeneous far field was done as inlet, outlet, shear-wall, and airfoil. The cell zone named ‘airfoil’ is a stationary wall with no slip condition with a standard roughness, these conditions are also applied for the shear-wall with the same constant sand-grain roughness of 0.5. The inlet has been set as
velocity inlet (velocity=29.5 m/s) with a direction set as normal to boundary. The outlet is a pressure outlet with a backflow reference frame set to absolute and the backflow direction as normal to boundary. The boundary conditions were applied to the meshed model with 1-equation viscous model Spalart-Allmaras and water as the medium.

Improved-

The boundary conditions given in the above section are to validate the model and meshes designed. In addition to that some more iterations with changed boundary conditions are done to get better results. It was seen that a variation of 1m/s from the base value of 29.5 m/s (28.5 to 31.5) gave a series of unique solutions for the designed model and mesh. Study of the obtained data was done keeping the purpose of a turbine in mind.

Table 1. Reference values and numerical simulation values

| Velocity | Reference Values | Numerical Analysis |
|----------|------------------|--------------------|
| LE       | Pressure        | PTE - Pressure     | LE       | Pressure        | PTE - Pressure     |
| PLE      | 28.5            | 1.44E+06           | -        | 6.48E+05       | -4.90E+05          |
| PTE      | 7.87E+05        | -                  | 6.17E+05 | 1.62E+06       | 4.90E+05           |
| LE       | 29.5            | 1.46E+06           | -        | 6.59E+05       | -7.40E+05          |
| PLE      | 8.01E+05        | -                  | 6.59E+05 | 1.70E+06       | -7.40E+05          |
| PTE      | 2.44E+06        | 2.44E+06           | 6        | 1.79E+06       | -7.90E+05          |
| LE       | 30.5            | 1.48E+06           | -        | 6.70E+05       | -7.90E+05          |
| PLE      | 8.14E+05        | -                  | 6.70E+05 | 1.79E+06       | -7.90E+05          |
| PTE      | 2.58E+06        | 2.58E+06           | 6        | 1.79E+06       | -7.90E+05          |
| LE       | 31.5            | 1.51E+06           | -        | 6.81E+05       | -8.30E+05          |
| PLE      | 8.28E+05        | -                  | 6.81E+05 | 1.88E+06       | -8.30E+05          |
| PTE      | 2.71E+06        | 2.71E+06           | 6        | 1.88E+06       | -8.30E+05          |

Figure 5: Residual Plot for ‘n+1’th iteration in ANSYS Fluent

4. Results and Discussion

A grid independence check was carried out with the same boundary conditions as described in the previous section. The number of elements and nodes were increased for several iterations by adding refinement to the discretization and by decreasing the edge sizing, making the mesh even finer.

Results were noted post every iteration. An ‘n+1’th iteration showed nearly same results (figure 5) as the ‘n’th iteration suggesting a successful grid independence check.

Table 2 shows the last 5 modifications in the element sizing and the corresponding values of maximum velocity over the airfoil.
Table 2: Iterations for Grid Independence Check

| Element Size | Velocity |
|--------------|----------|
| 7.00E-03     | 86.302   |
| 7.10E-03     | 84.993   |
| 7.50E-03     | 85.851   |
| 8.00E-03     | 85.750   |
| 8.80E-03     | 85.749   |

The contours for different inlet velocities were studied. Figures 6, 7 show contours for inlet velocity of 29.5 m/s.

Results for 29.5m/s inlet velocity:
The velocity at the leading edge was found out to be 26.21 m/s, and that at the trailing edge 43.68 m/s and the maximum velocity over the airfoil was 83 m/s. The pressure at leading was seen to be 2.44e6 Pa, 1.7e6 Pa at trailing edge and 2.77e3 Pa at mid-chord.

Results for 28.5 m/s inlet velocity: (figures 8, 9)
The velocities at the leading edge, trailing edge, and mid-chord are 17.5 m/s, 39.37 m/s and 74.37 m/s respectively and the corresponding static pressure values are 2.31e6 Pa, 1.62e6 Pa, 3.502e4 Pa.

Results for inlet velocity of 30.5m/s: (see figures 10, 11)
The velocities at the leading edge, trailing edge, and mid-chord are 20.52 m/s, 41.04 m/s and 82.09 m/s respectively and the corresponding static pressure values are 2.58e6 Pa, 1.79e6 Pa, 1.16e4 Pa.

Results for inlet velocity of 31.5m/s: (see figures 12, 13)
The velocities at the leading edge, trailing edge, and mid-chord are 26.5 m/s, 42.39 m/s and 84.79 m/s respectively and the corresponding static pressure values are 2.71e6 Pa, 1.88e6 Pa, 2.13e5 Pa.

The pressure values at leading and trailing edge were tabulated (Table 1) to show variation in performance characteristics of the blade.

![Image of velocity contour](https://via.placeholder.com/150)

Figure 6. Velocity Contour for 29.5m/s inlet
Figure 7: Static Pressure Contour for 29.5m/s inlet

Figure 8: Velocity Contour for inlet of 28.5m/s

Figure 9: Contours for Static Pressure at an inlet velocity of 28.5m/s
Figure 10: Contours of Static Pressure for 30.5 m/s inlet

Figure 11: Velocity Contour for 30.5 m/s inlet

Figure 12: Static Pressure Contour for 31.5 m/s inlet
Figure 13: Velocity Contour for 31.5 m/s inlet

Figure 14: Performance Comparison of Numerical Analysis and Reference Data based on Table 1

5. Conclusions

In order to improve the pressure ratio of a LO$_X$ turbo-pump in an SSME, a numerical analysis was carried out on a 2D model of the turbine rotor blade on ANSYS Fluent. Appropriate fluent settings and suitable boundary conditions were applied.

Results obtained were validated with an allowed tolerance of less than 10%. Increase in entry velocity was the choice of further improvement, a sharp increase in the pressure ratio of the turbine rotor blade was observed. With an increase in inlet velocity better results were observed. Finally, at an inlet velocity of 31.5 m/s the pressure difference of 8.3e5 (figure 14) across the rotor blade was considered to be the best result.

Acknowledgements

We are greatly indebted to Professor Srinivas G. for guiding us towards the completion of this article.

References

[1] D.G. Ainley, G.C.R. Mathieson, *A method of performance estimation for axial flow turbines*, Aero Research Council Reports and Memoranda 2974, Her Majesty’s Stationery Office, London, 1951
[2] J. Boynton, H. Rohlik, *Effect of tip clearance on performance of small axial hydraulic turbine*, Lewis Research Centre, Cleveland, Ohio 44135, 1976, NASA TM X-3339.

[3] B. Lakshminarayana, J.H. Horlock, *Tip-Clearance Flow and Losses for an Isolated Compressor Blade*, Research Council Reports and Memoranda, n.3316, Ministry of Aviation Aeronautical, London, 1962, 24 p. (replaces A.R.C.23.818)

[4] C.H. Sieverding, Recent progress in the understanding of basic aspects of secondary flow in turbine blade passage, *J. Eng. Gas Turbines Power* **107** (April 1985) 248-257.

[5] S. Sjolander, K. K. Amrud, *Effects of tip clearance on blade loading in a planar cascade of turbine blades*, ASME Paper Number 86-GT-245, 1986.

[6] A.K. Saha, Blade tip leakage flow and heat transfer with pressure-side winglet, *Int. J. Rotating Mach.* (2006), 15 p.

[7] C.M. Rhie, *Numerical study of the turbulent flow past an isolated airfoil with trailing edge separation*, AIAA Paper 82-0998, 1982.

[8] J. Dunham, Improvements to the Ainley-Mathieson method of turbine performance prediction, *J. Eng. Power* **92** (July 1970) 252.

[9] S.C. Kacker A mean line prediction method for axial flow turbine efficiency, *J. Eng. Power* **104 (1)** (1982) 111-119.

[10] Luiz Henrique L. W. Jesuino T. T. Cleverson Bringhenti. An evaluation of the tip clearance effects on turbine efficiency for space propulsion applications considering liquid rocket engine using turbo-pumps, *Aerospace Science and Technology* **70** (2017) 55-65.

[11] A. Richard Felix James C. Emery *A comparison of typical NGTE and NACA axial-flow compressor blade sections in cascade at low speed*, NATIONAL ADVISORY COMMITTEE FOR AERONAUTICS, Technical Note 3937, 46 p. (March 1957)

[12] https://www.pinterest.com/pin/517843657152926425/

[13] Srinivas G, Madhu Gowda B P “Aerodynamic Performance Comparison of Airfoils by Varying Angle of Attack Using Fluent and Gambit” *Journal of advanced materials research*, vol. 592-594, 2014. pp 1889-1896.

[14] Aditya, A., & Srinivas, G. (2019). The numerical analysis of NACA 0018 airfoil: Studying the effect of flap. *International Journal of Mechanical and Production Engineering Research and Development*, **9**(4), 1047-1054.