# A Parametric Approach to Turbine Tip Leakage Aerodynamic Investigation for Axial Flow Turbine

Levent Kavurmacioglu, Hidir Maral, Cem Berk Senel

# A Parametric Approach to Turbine Tip Leakage Aerodynamic Investigation for Axial Flow Turbine

Levent Kavurmacioglu[1]*, Hidir Maral[1], Cem Berk Senel[1]

**ISROMAC 2016**

**International Symposium on Transport Phenomena and Dynamics of Rotating Machinery**

**Hawaii, Honolulu**

**April 10-15, 2016**

**Abstract**

This current paper deals with the development of a computational system dealing with parametric assessment of aerodynamic losses near the tip region of a high pressure turbine blade. Performance of an axial flow turbine is strongly related to the flow structure in the tip gap between blade tip and casing. Flow structure in the tip gap is a significant source of inefficiency.

Special emphasis is paid to developing a sufficiently accurate 3D RANS based loss estimation system for the optimization of tip section geometry. An HP turbine tip section will be optimized for minimum aerodynamic losses and also for minimized heat loss. The present study deals with a preliminary study of effective parametric grid generation and turbulent flow model implementation and assessment under realistically simulated turbine flow conditions. Initial development of this computational model is performed in a linear turbine cascade arrangement.

Numerical experiments with parametrically generated multizone structured grid topologies and unstructured grids pave the way for the 3D optimization of the HP turbine blade tip region. The future efforts will include modified squealer tips, tip trenches and tip carving investigations in an effort to obtain optimal 3D tip shapes as far as aerodynamic losses and heat losses are concerned.

**Keywords**

Axial Flow Turbine — Linear Cascade — CFD — Parametric Study — Tip Leakage

[1] *Department of Mechanical Engineering, Istanbul Technical University, Istanbul, Turkey*

***Corresponding author**: kavurmacio@itu.edu.tr,

## INTRODUCTION

In turbomachinery field to allow relative motion of blades and to protect the blade surface which is exposed to hot gas stream a gap is required. Performance of turbomachine is strongly related to the flow structure within this gap and flow in this gap is 3-D and highly complex. A pressure driven flow between pressure side and suction side of the blade throughout the gap results in approximately one-third of the aerodynamic loss in rotor of axial gas turbine [1]. The leakage flow generally rolls into vortical structure near the suction side of the blade. Leakage flow, tip vortex and interaction of these flow structures with secondary flow, endwall boundary layer and main stream flow reduce the aerodynamic performance of the turbomachine. Besides, a reduction in work output of the turbine is observed since the flow passes over the blade tip is not turned as the main flow [1-3]. Leakage flow is also a significant source of higher thermal loads at blade tip [4].

Many studies have been carried out in order to clarify structure of leakage flow and reduce the effects of it. Unshrouded turbines have been studied intensively compared to shrouded turbines since shrouded turbines bring difficulties about both mechanical and thermal loadings. Moore and Tilton investigated the leakage flow analytically and experimentally by assuming the leakage flow as a flow through an orifice [5]. Morphis and Bindon investigated the effects of blade tip geometry and radius of pressure side corner on aerodynamic loss [6]. Heyes et all obtained that squealer tip geometries could reduce the leakage flow as tip gap is blocked by separation bubble [2]. Experiments by Yaras and Slojander showed that leakage flow rate was decreased by the effect of moving casing that modeled the relative motion between blades and casing [7]. Secondary flow near the endwall of planar cascade was visualized using multiple smoke wires by Wang et al. [8].

Passive control methods are the most common approach in order to reduce the loss related leakage flow. Squealer, partial squealer and winglet blade tip geometries are widely used. Mischo et al. showed that performance of an axial flow turbine could be improved by confining the leakage flow within the cavity of a squealer tip geometry [1]. Heyes et al. observed that application of squealer geometry could reduce the tip leakage losses [2]. In a numerical study on the effect of squealer tip geometry on heat transfer and efficiency, Ameri et al. obtained that leakage flow rate was reduced whereas heat transfer to the blade tip was increased [9]. Camci and Dey found out that using tip platform extensions could affect the aerodynamic performance by weakening the tip vortex [10]. Measurements by Azad et al. indicated that implementation of suction side squealer had better results with respect to pressure side squealer [11]. An experimental study on heat transfer was carried out by Kwak et al. to determine the effect of the position of squealer [12]. Considerable reduction in heat transfer was achieved in a recent study that modeled the tip gap as convergent-divergent channel by Zhang and Le [13].

There have been less study about tip carving in the

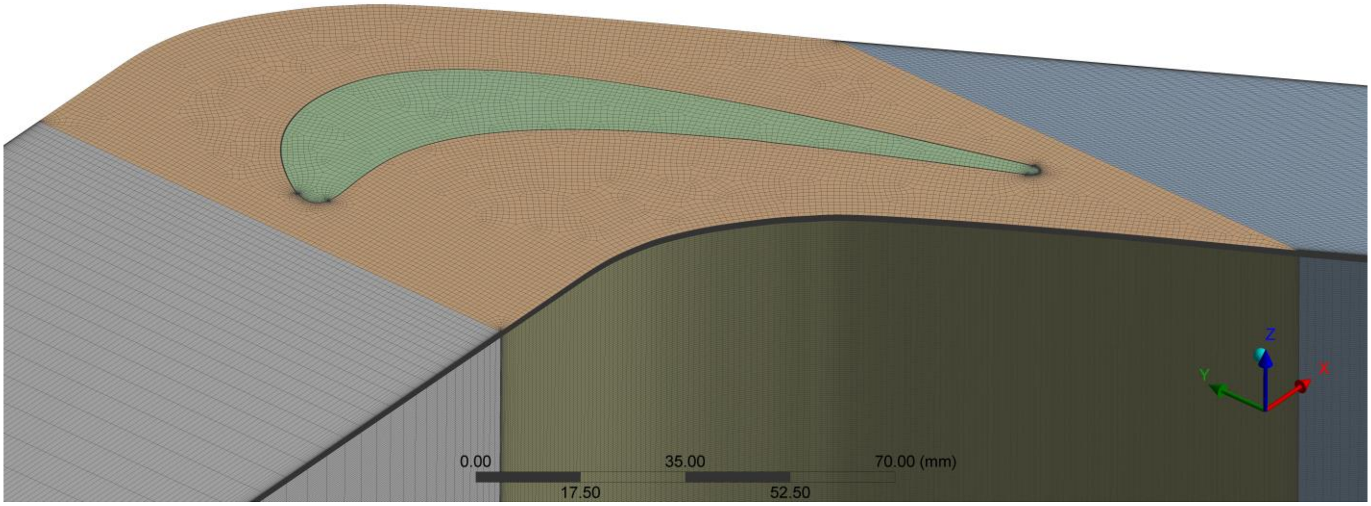



**Figure 1.** Grid generation obtained parametrically

literature. Maaesschalck et al. investigated the effect of tip shaping on aerodynamic performance of axial turbine [14]. In this paper, flow structure of leakage flow is tried to be investigated in a comprehensive way. A parametric study on the effect of tip gap height is going to be carried out numerically as seen in Figure 1. Aerodynamic loss related to the leakage flow will be determined for the tip clearance values of 0.7 %, 1.0 %, and 1.5 %. This parametric approach will pave the way for passive control methods and optimization process for tip shaping. Axial turbine blade tip profile used in calculations belongs to Pennsylvania State University Axial Flow Turbine Research Facility (AFTRF) [15].

## 1. METHODS

A numerical study has been carried out in this study. In the field of turbomachinery experimental measurements may become difficult, expensive and time consuming. In such circumstances Computational Fluid Dynamics (CFD) method takes an important role to understand the flow structure.

Parametric assessment of aerodynamic loss corresponds to the most important part of this study. Grid generation process has been carried out via ANSYS Workbench. Computational domain which comprised multiple blocks was obtained using SpaceClaim module in ANSYS. As seen from Figure 2, computational domain was modeled as a linear cascade arrangement for a single blade. Dimensions to obtain the flow field were defined parametrically on the interface of ANSYS Workbench. Computational domain was divided into multi blocks in order to provide a parametric definition and achieve a fully hexagonal grid in a simple way. Creating multi blocked flow domain enables to use multizone method in ANSYS Meshing module. Each of inlet and outlet blocks involve one block whereas rotor domain is divided into three blocks in this study. Rotor consists of basically two blocks; rotor-blade domain and gap domain as seen from Figure 2. Then gap domain is separated into 2 blocks to be able to control the mesh sizes to achieve the required $y^+$ values.

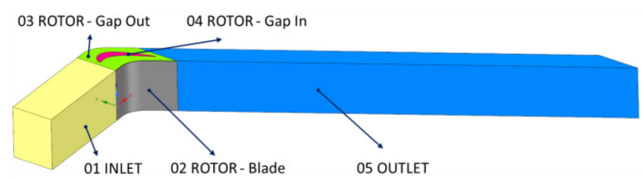



**Figure 2.** Computational domain obtained in a parametric way

Figure 3 shows the definition of the tip gap height in SpaceClaim. By setting up the dimensions in a parametric way, time required to form a new solid model has been reduced considerably. Tip clearance ratio has been introduced as a design parameter in terms of blade height in present study.

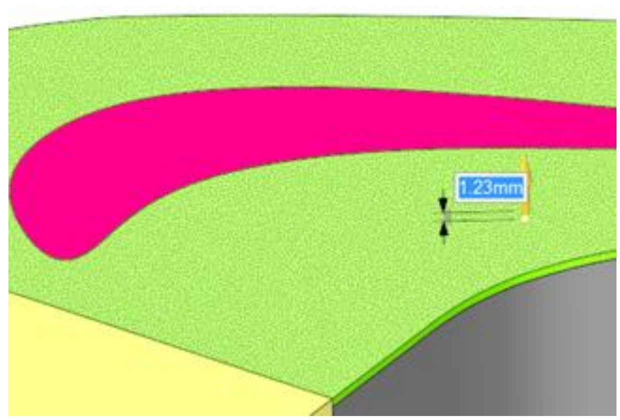


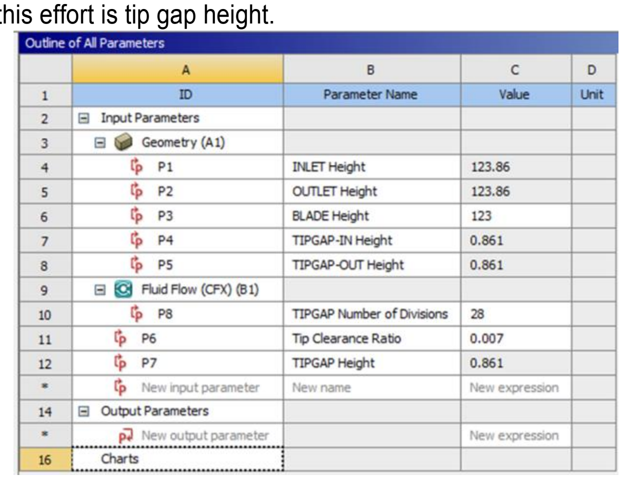

**Figure 3.** Definition of tip gap height.

Once the computational domain is formed, geometry is transferred to meshing module on ANSYS Workbench. ANSYS Meshing module was used to generate a structured mesh in a parametric assessment using multizone method. Basic variables to for mesh generation can also be introduced as parameters. Similar to procedure in solid modelling, grid sizing functions were defined parametrically. Number of division in the tip gap was introduced as a parameter different from all other definitions in meshing process. Application of parametric approach in grid generation enabled an efficient working environment in CFD calculations.

Afterwards, the mesh file was transferred to CFX-Pre for modeling and then CFX Solver for the simulation. Mass flow rate and static pressure boundary conditions have been imposed at inlet and exit respectively. Turbulence intensity and length scale at inlet section were specified as 0.5% and 0.123 m. Boundary conditions represent the realistic conditions of AFTRF. A comparison to experimental results are not available since AFTRF is a rotational test set-up. Also, blade tip geometry has been used, not solid model of the blade itself.

Considering the number of cases for CFD methods, solid modelling and grid generation becomes significant productivity issues. Parametric study removes the necessity of modeling of the computational steps at each time. Grid generation corresponds to the most problematic step in numerical calculations. It may take a long time for a single model. Within this study we experienced that parametric approach reduced time at each step for numeric calculation. Parametric assessment reduced the required time for a complete numerical solution to one-sixth excluding simulation time.

Parametric definition is illustrated in Figure 4. Basic parameters for current numerical study are tip clearance ratio in terms of blade height and number of division in tip gap. The rest of the parameters given under geometry part have been defined depending on the clearance ratio. Basic difference among the cases investigated within the scope of this effort is tip gap height.



Outline of All Parameters

| | A | B | C | D |
|---|---|---|---|---|
| 1 | ID | Parameter Name | Value | Unit |
| 2 | Input Parameters | | | |
| 3 | Geometry (A1) | | | |
| 4 | P1 | INLET Height | 123.86 | |
| 5 | P2 | OUTLET Height | 123.86 | |
| 6 | P3 | BLADE Height | 123 | |
| 7 | P4 | TIPGAP-IN Height | 0.861 | |
| 8 | P5 | TIPGAP-OUT Height | 0.861 | |
| 9 | Fluid Flow (CFX) (B1) | | | |
| 10 | P8 | TIPGAP Number of Divisions | 28 | |
| 11 | P6 | Tip Clearance Ratio | 0.007 | |
| 12 | P7 | TIPGAP Height | 0.861 | |
| * | New input parameter | New name | New expression | |
| 14 | Output Parameters | | | |
| * | New output parameter | | New expression | |
| 16 | Charts | | | |

**Figure 4.** Parametric definitions on Workbench interface.

Tip clearance is defined in term of blade height; ratio of tip gap height to blade height. Three different tip clearance cases 0.7% (FLAT0007) 1.0% (FLAT0010), and 1.5% (FLAT0015), are going to be investigated.

### 1.1 Numerical Method

All simulations in present study were carried out using commercial software ANSYS as pointed out previously. SpaceClaim module was used for modeling of computational domain, meshing module for grid generation and CFX for the numerical calculations. Computations were performed for a single blade with periodic boundary conditions in pitchwise direction.

Blade tip geometry was used to create solid model of the axial turbine. Profile at blade tip was extruded to obtain a cylindrical blade. Specifications of the blade geometry was given in table 1. Length of inlet domain is equal to 3 axial chord length whereas it is equal to 6 axial chord for outlet domain in order to reduce gradients in streamwise direction. Reynolds number based on inlet velocity and axial chord length is calculated as 143500

*Table 1: Blade specifications*

| | |
|---|---|
| *Blade height [mm]* | 123 |
| *Axial chord [mm]* | 85.04 |
| *Flow angle (°) (inlet)* | 71.3 |
| *Pitch [mm]* | 99.274 |

Multizone method was used for grid generation. Multizone, which is a type of blocking approach similar to ICEM CFD, uses automated topology decomposition and generates structured hexa mesh where blocking topology is available [16]. Fully hexagonal elements have been used in calculations in order to reduce solution time and increase the accuracy. In order to resolve the boundary layer effects inflation (boundary layer mesh) was applied. Grid around rotor domain is seen in Figure 1.

The 3D incompressible RANS equations were solved using a finite volume discretization with the assumption of steady state flow at 25°C.

A two equation turbulence model was used in the calculations. Shear Stress Transport (SST) turbulence model was performed. SST model is a combination of standard (k-ε) and (k-ω) models to overcome the shortcomings of each model by a blending function depending on the distance away from the wall [17]. For using SST model y+ value should be less than 2, in some resources 1. In present study for each tip clearance heights this condition was satisfied; y+<2. Number of elements in the cases are approximately 14 million. This high numbers are due to number of elements in tip gap region to resolve the flow structures accurately and to reduce aspect ratio to an acceptable level. Initial height of the first cell away from the wall should be as small as to keep y+ values around 1, which results in higher aspect ratios.

## 2. RESULTS AND DISCUSSION

### 2.1 Results and Discussion

In this section numerical results are going to be given in details with flow visualizations in different sections of the computational domain.

In numerical calculations grid dependency study is an important issue to deal with. More importantly, $y^+$ is a crucial measure for boundary layer. Firstly, a grid dependency study has been performed including coarse, medium and fine cases. Number of elements at each case and number of division in tip gap are given in Table 2. Tip gap height has been divided into 30, 35, and 40 respectively for COARSE, MEDIUM and FINE cases.

*Table 2: Mesh specifications for grid dependency*

| Cases | Number of Elements | Number of Divisions in Tip Gap |
|---|---|---|
| **COARSE** | 13,208,854 | 30 |
| **MEDIUM** | 13,593,447 | 35 |
| **FINE** | 14,002,526 | 40 |

Distribution of static pressure coefficient has been examined for grid dependency study. Figure 5 shows static pressure coefficient, $C_P$, distribution near blade tip. $C_P$ is defined as

$$C_p = \frac{P - P_{ref}}{\frac{1}{2}\rho U_m^2} \qquad (1)$$

$P_{ref}$ is the area averaged pressure at inlet plane, and $U_m$ reference velocity. Reference velocity corresponds to exact value of test rig.

Figure 5 indicates that there are small differences between three cases whereas MEDIUM and FINE $C_P$ curves are closer to each other compared to COARSE case. FINE and MEDIUM cases differ slightly from each other. Considering the highly complex flow structure in tip gap, blade tip becomes a critical region for mesh dependency study. Thus, near blade tip region has been used instead of blade tip itself.

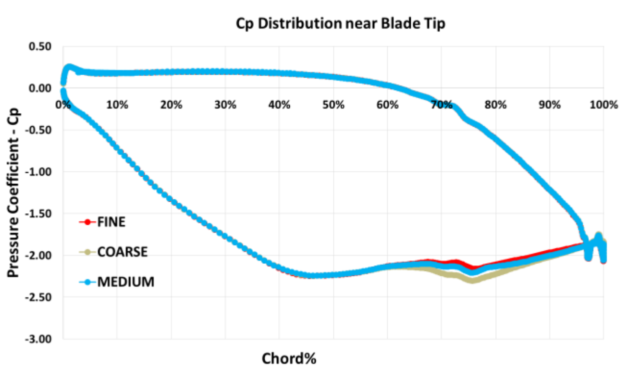



**Figure 5:** Mesh dependency study for FLAT0010

In numerical studies $y^+$ is considered as an important issue. Height of first cell from wall in non-dimensional form is defined as $y^+$. In order to resolve the flow within the boundary layer, it should be kept as small as. Flow through the tip gap is highly complex and turbulent, thus order of $y^+$ becomes very important to investigate the flow field. Also, for turbulence modeling, order of $y^+$ is a necessary condition. In present study order of $y^+$ is plotted at two different spanwise terminals including midpspan and near blade tip. Figure 6 and Figure 7 indicate that $y^+$ condition is satisfied. Maximum $y^+$ value is 0.27 and average 0.15 for all cases at midspan. Besides, $y^+$ values around the blade profile is much smaller than 2.

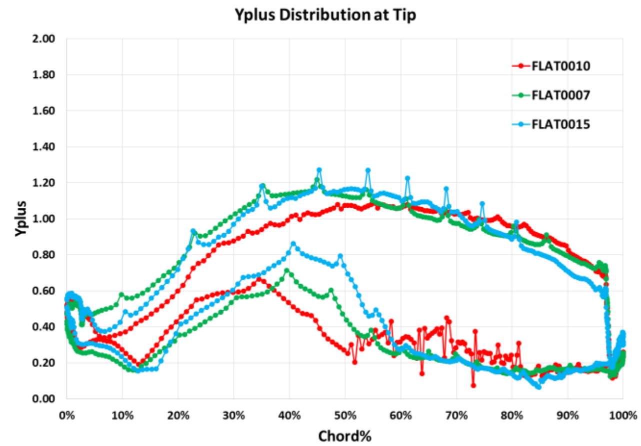



**Figure 6**. $y^+$ distribution at blade tip.

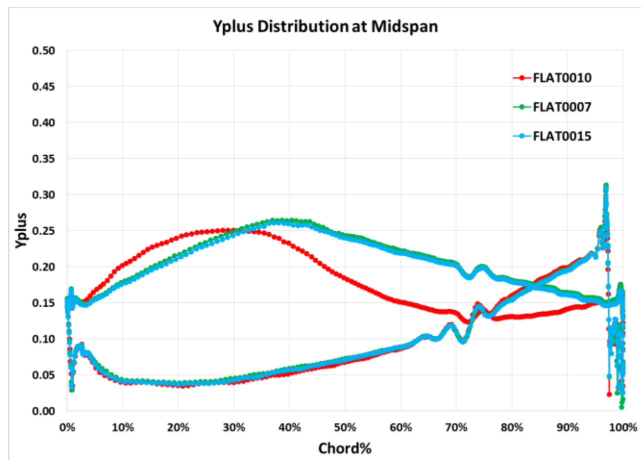



**Figure 7**. $y^+$ distribution at midspan.

Blade tip region is a critical region due to high shear effects to ensure recommended $y^+$ level. As seen from Figure 6, maximum value is around 1.3 which means $y^+$ is kept at reasonable level. Average value for $y^+$ is around 0.5 for all cases. Less than 1.3 is achieved at lower spanwise stations.

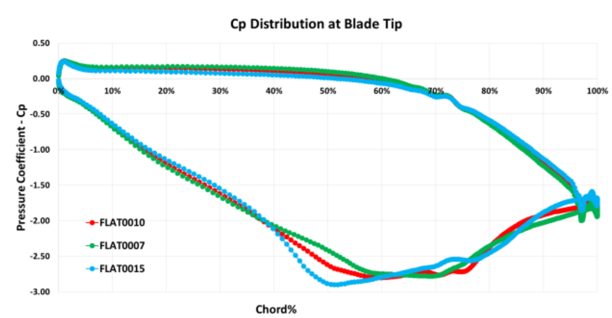



**Figure 8**. $C_P$ distribution at blade tip

Figure 8 shows static pressure coefficient ($C_P$) distribution at blade tip along streamwise direction. It can be concluded that minimum pressure is at case FLAT0015. Besides, figure 8 indicates that blade loading is higher for the highest tip clearance.

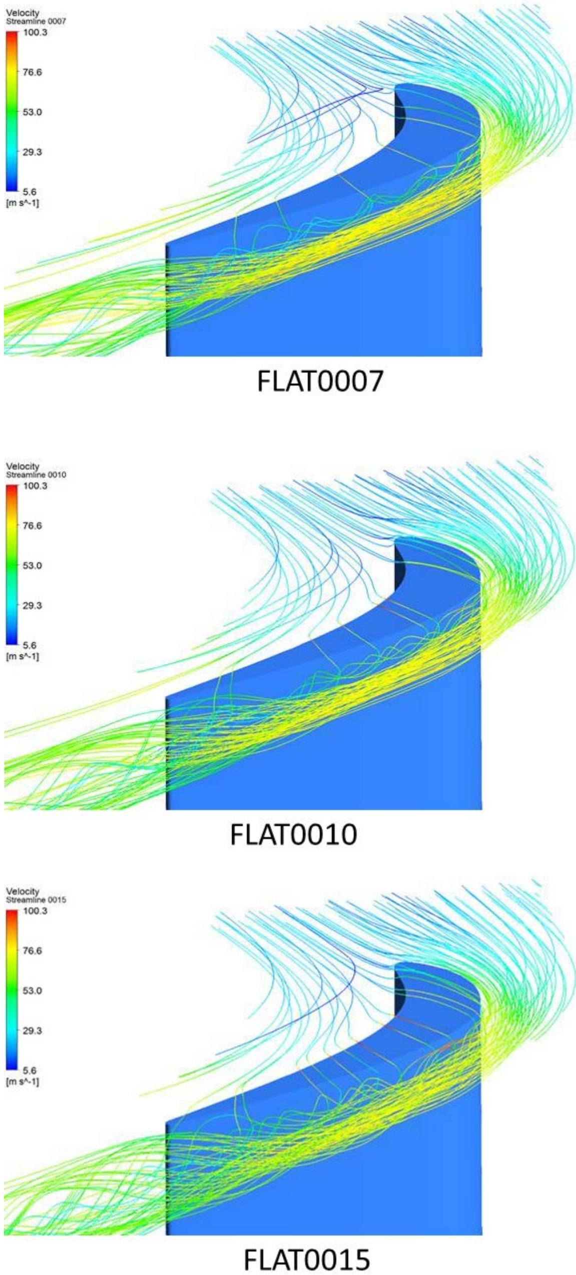



**Figure 9**. Velocity streamlines in the tip gap.

For flow visualization streamlines are widely used to provide a better understanding. Streamlines released upstream of the blade leading edge in the vicinity of tip clearance visualize the formation of tip leakage vortex. Figure 9 depicts how leakage flow rolls into tip vortex near suction side of the blade. Pressure driven flow passes over the blade tip tends to evolve into a vortical structure. Formation of tip vortex verifies that flow becomes highly complex near suction side. Streamlines in Figure 9 indicates intensity of shear region near suction side increases. Increase in tip gap height results in larger tip vortex. Thus total pressure loss due to highly shear region is expected to be higher for larger tip clearances.

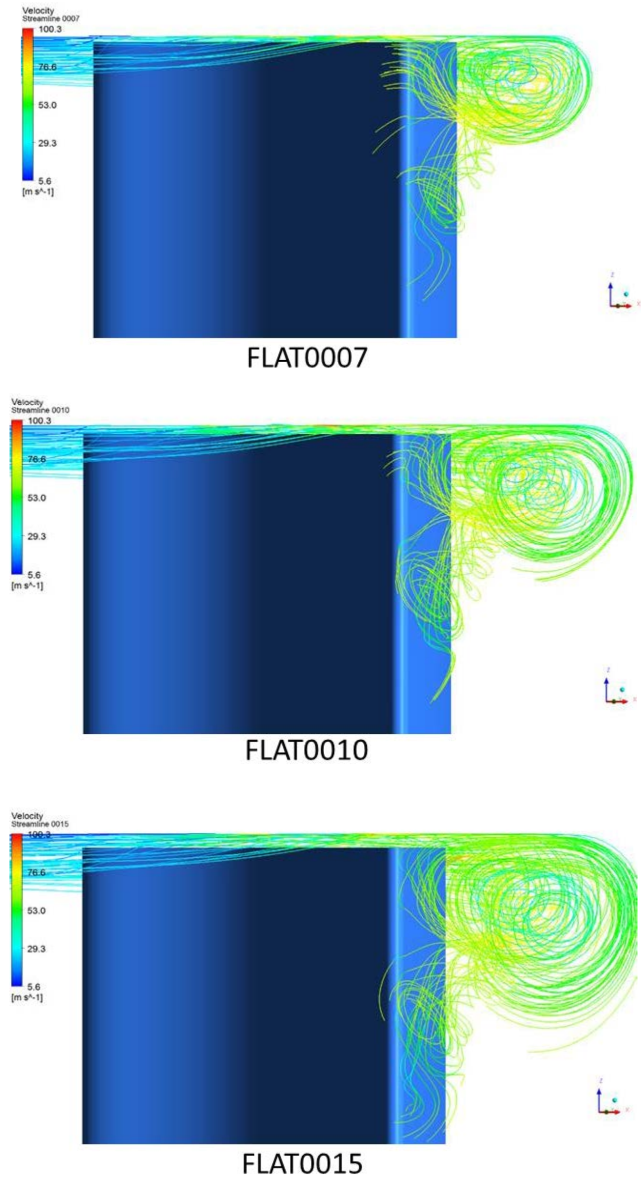



**Figure 10**: Tip vortex visualization near suction side.

Effect of tip gap height on tip vortex is visualized near suction side of the blade using streamlines. Figure 10 provides a better understanding to see the influence of tip clearance. Tip leakage flow evolves into a larger tip vortex for the case FLAT0015. Obviously seen that tip vortex is dominant to passage vortex. A weak passage formation is valid for all three tip clearances. Total pressure distribution at exit plane confirms this phenomena. It can be concluded that tip vortex suppresses passage vortex. For larger tip gap heights, tip vortex spreads a wider region as streamlines released upstream of leading edge apparently. Influence area of tip vortex is extended to lower spanwise locations.

Leakage flow rate is an important criteria to determine the aerodynamic loss. Reducing leakage flow rate is one of the main goals to improve the aerodynamic performance. Thus, tip gap height is tried to be smaller as possible as. However, there is a physical limit to avoid mechanical problems. Flow rate along streamwise direction is shown in Figure 11. Leakage flow distribution exhibits similar trends for all cases. At same axial location, flow rate takes the smallest value for

FLAT0007 and largest value for FLAT0015. Leakage flow rate reaches its highest value around 80% of axial chord for all cases.

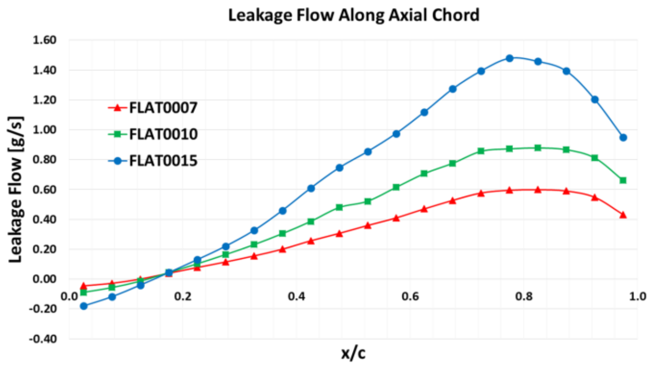



**Figure 11.** Leakage flow rate along streamwise direction

Determination of location where leakage flow becomes larger can be used in passive control methods to reduce the flow rate.

Total pressure distribution is another important variable for comparing aerodynamic performance. Total pressure distribution at exit plane is going to be used to calculate aerodynamic loss. Exit plane is placed 1.05 axial chord distance from the leading edge to be perpendicular to the camber line of the blade.

Total pressure contour (non-dimensional) downstream of the trailing edge is a comprehensive tool for predicting the flow structures from hub to tip. Figure 12 indicates tip vortex, passage vortex just beneath the tip vortex, endwall effects and hub vortex. In all three cases tip vortex is dominant to the passage vortex. Vortical structure near hub region is similar to each other. In addition, it can be concluded that end wall losses are at the same order of magnitude. However, tip vortex is different for all cases. Tip vortex is weaker for case FLAT0007 compared to other two cases. From figure it is obvious that increasing tip clearance, strengthens the tip vortex. Tip vortex becomes larger for the highest tip gap and expands to a wide region in pitchwise direction. It covers almost the whole region up to periodic side for case FLAT0015.

Total pressure loss coefficient, $\Delta C_{p0}$, was defined as the difference between mass flow averaged total pressure coefficients at exit and inlet planes. Total pressure coefficient, $C_{p0}$, was defined similar to static pressure coefficient, $C_p$.

$$C_{p0} = \frac{P_0 - P_{0,ref}}{\frac{1}{2}\rho U_m^2} \tag{2}$$

$$\Delta C_{p0} = C_{p0,exit} - C_{p0,inlet} \tag{3}$$

$\Delta C_{p0}$, was calculated for each case as seen in Figure 12. Then, the results were recalculated with reference to 1 % case. Calculations indicate that reducing tip gap height results in decrease in total pressure loss which means improvement in aerodynamic performance. Total pressure loss is determined (-4.28)% for tight gap (FLAT0007) with respect to base (FLAT0010) and 11.71% for larger gap.

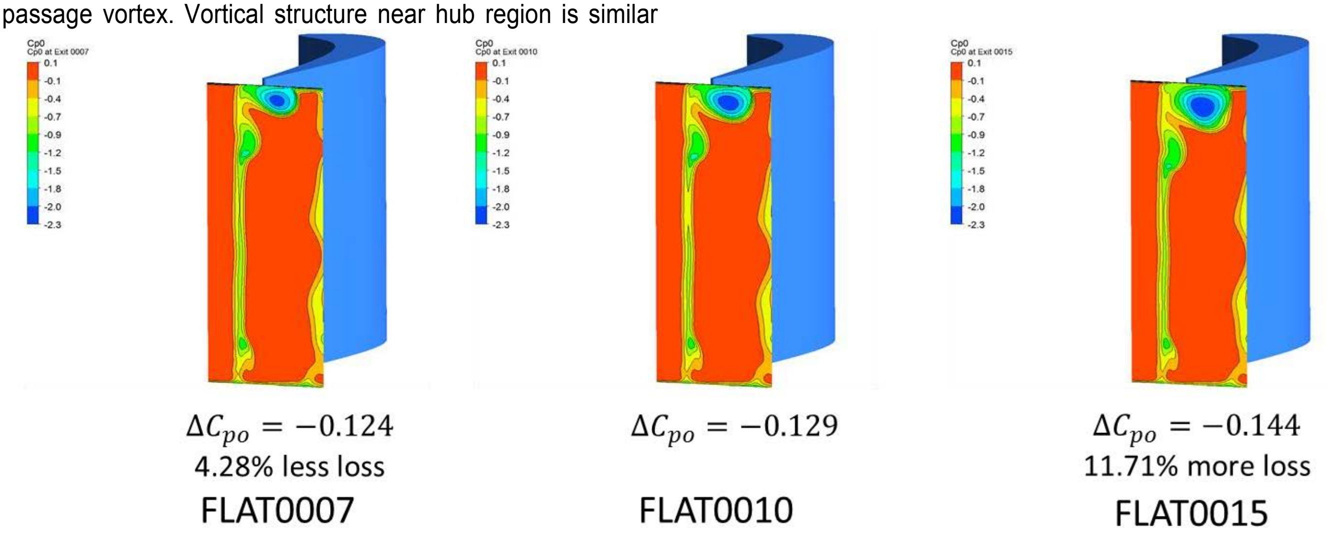



**Figure 12.** Total pressure distribution at exit plane and aerodynamic loss.

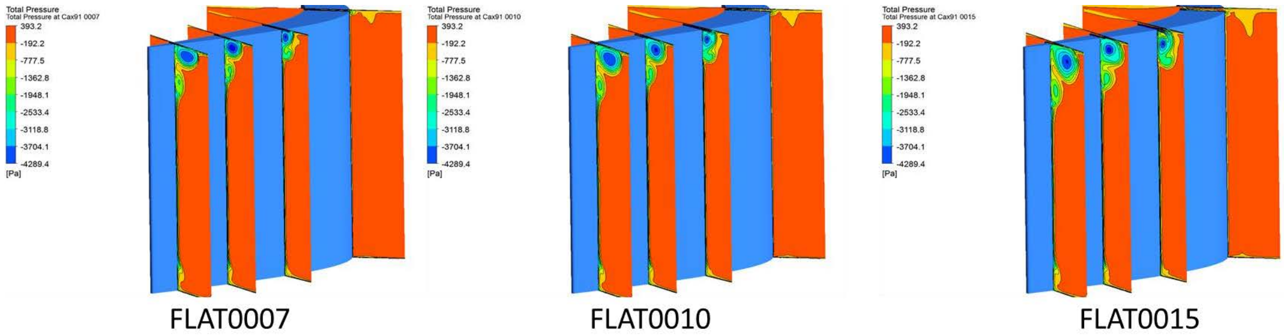



**Figure 13**: Total pressure contour along streamwise direction.

Figure 13 shows total pressure contours along chordwise direction. It can be deduced from figure that formation of tip vortex is observed after 20 % of axial chord. Besides, evolution of tip vortex throughout the streamwise direction at different axial terminals is demonstrated. Total pressure contour enables to make further suggestions about passive control methods to reduce the loss related to the leakage flow.

## 2.2 Conclusion

A numerical prediction for flow in tip gap was presented. A detailed investigation has been carried out to make the flow structure in tip gap clear.

The remarkable point in present study is to form computational domain and generate mesh by introducing a parametric approach. Parametric approach has reduced the time considerably for numerical calculations. Thus, it paves an efficient method in CFD.

Results from numerical calculations consistent with previous studies in literature. Increase in tip clearance height leads to an increase in total pressure loss since tip vortex becomes effective almost the whole region up to the periodic surface. Leakage flow rate shows tendency to increase for higher values of tip gap height. Effect of passage vortex is very small with respect to tip vortex. Total pressure contour and leakage flow distribution are useful tools to offer new designs. In future studies to reduce the aerodynamic loss these results will be used.

## ACKNOWLEDGMENTS

The authors grateful to Prof. Cengiz Camci for his great support and contriubution. Prof. Camci allowed to use AFTRF blade tip geometry in present study.

## REFERENCES

[1] B. Mischo, T. Behr, R. S. Ahbari. Flow physics and profiling of recessed blade tips: Impact on performance and heat load. *Journal of Turbomachinery*, 130(2), 2008

[2] F. J. G. Heyes, H. P. Hodson, and G. M. Dailey. The effect of blade tip geometry on the tip leakage flow in axial turbine cascades. *Journal of Turbomachinery*, 114(3): 643-651, 1992

[3] S. K. Krishnababu, P. J. Newton, W. N. Dawes, G. D. Lock, H. P. Hodson, J. Hannis, and C. Whitney. Aerothermal investigations of tip leakage flow in axial flow turbines-Part I: effect of tip geometry and tip clearance gap. *Journal of Turbomachinery*, 131(1), 2008.

[4] N. L. Key,T. Arts. Comparison of turbine tip leakage flow for flat tip and squealer tip geometries at high-speed conditions. *Journal of Turbomachinery*, 128(2): 213-220, 2004.

[5] J. Moore, J. S. Tilton. Tip leakage flow in a linear turbine cascade. *Journal of Turbomachinery*, 110(1): 18-26, 1988.

[6] J. P. Bindon, G. Morphis. Development of axial turbine leakage loss for two profiled tip geometries using linear cascade data. *Journal of Turbomachinery*, 114(1): 198-203, 1992.

[7] M. I. Yaras, S. A. Sjolander. Effects of simulated rotation on tip leakage in a planar cascade of turbine blades: Part I-tip gap flow. *Journal of Turbomachinery*, 114(3): 652-659, 1992

[8] H. P. Wang, S. J. Olson, R. J. Goldstein and E. R. G. Eckert. Flow visualization in a linear cascade of high performance turbine blades. *Journal of Turbomachinery*, 119(1): 1-8, 1997.

[9] A. A. Ameri, E. Steinthorsson, D. L. Rigby. Effect of squealer tip on rotor heat transfer and efficiency. *Journal of Turbomachinery*, 120(4): 753-759, 1998.

[10] C. Camci, D. Dey. Aerodynamic tip desensitization of an axial turbine rotor using tip platform extensions. *ASME Turbo Expo: Power for Land, Sea and Air*, New Orleans, Louisiana, USA, 2001.

[11] G. S. Azad, J.-C. Han, R. S. Bunker, C. P. Lee. Effect of squealer geometry arrangement on a gas turbine blade tip heat transfer. *Journal of Heat Transfer*, 124(3): 452-459, 2002.

[12] J. S. Kwak, J. Ahn, J. –C. Han, C. P. lee, R. S. Bunker, R. Boyle, R. Guagler. Heat transfer coefficients on the squealer tip and near-tip regions of a gas turbine blade with single or double squealer. *Journal of Turbomachinery*, 125(4): 778-787, 2003.

[13] Q. Zhang, L. He. Tip shaping for HP turbine blade aero-thermal performance management. *ASME Turbo Expo: Turbine Technical Conference and Expoisition* , 2012.

[14] C. De Maesschalck, S. Lavognoli, G. Paniagua, and T. Verstraete. Blade tip carving effects on the aerothermal performance of a transonic turbine. *Journal of Turbomachinery*, 137(2), 2015.

[15] Zaccaria, M. A. An experimental investigation into the steady and unsteady flow field in an axial flow turbine. PhD Thesis, The Pennsylvania State University, 1994.

[16] ANSYS Meshi-Intro_16.0_L04_Meshing Methods.

[17] ANSYS CFX Intro_16_L10 Turbulence