CFD Study to Analyze the Behavior of Secondary Flow Using Fillets

Lourdes Tafur Bermúdez
Master of Engineering in Mechanical Engineering
Bernardo Restrepo, Ph.D.
Mechanical Engineering Department
Polytechnic University of Puerto Rico

Abstract — *Secondary flow are a series of vortex created at the leading edge of the blade. These vortexes decrease the efficiency of the turbine by creating a non-uniform flow which will pass to successive blade stages. The secondary flow also causes heat transfer problems by moving hot air from the center of the passage to the endwall. Some methods are being studied to weak these vortexes including adding fillets to the leading edge of the blade. For this article a CFD study was perform using four different fillets to examine the behavior of the vortex by including the fillets. The design fillets were not able to eliminate the vortex and flow separation but the tall fillets seemed to increase the velocity at the passages. While the small area fillets seemed to worsen the flow separation.*

Key Terms— *CFD, Horseshoe Vortex, Secondary Flow, Turbine.*

INTRODUCTION

Gas turbine engines are under continues research to improve the overall efficiency. Improving the efficiency will translate in less consumption of fuel and reducing cost. Gas turbines engines have three basic components: compressor, combustor and turbine. First the compressor increases the total energy of the incoming fluid by compressing the fluid. The fluid then moves to the combustor where it will be combined with the fuel and then ignited to continue increasing the fluid energy. For the last stage, the turbine will expand out the fluid releasing the energy back to the engine by causing the turbine's blade to rotate due to the pressure drop across the blades.

Secondary Flow

The turbine is a component that has many ways in which its efficiency could be increased. One way will be the reduction of secondary flow. Secondary flows are a series of vortex created at the blade endwall junction. The first vortex is called horseshoe vortex. This vortex is created at the leading edge of the blade and the endwall junction. This vortex then divides toward the pressure side and the suction side of the blade as seen in figure 1 and 2 [1]. The pressure side vortex moves towards the adjacent blade's suction surfaces meeting the suction side vortex and forming the passage vortex. This vortex is then strengthened by the crossflow which is the flow created by the pressure gradients of the blades [2].

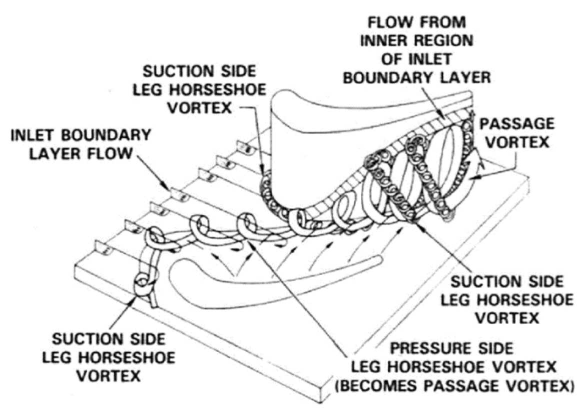

Figure 1
Secondary Flow Illustration

The secondary flow creates aerodynamic losses by creating a less uniform exit flow field for successive blade stages decreasing the efficiency. These vortices also cause heat transfer problems by moving hot fluids from the center of the passage to the endwall. The outlet temperature of the combustor is then restricted to avoid burning and melting the turbine components. By improving the heat transfer and avoiding additional pressure and aerodynamics losses the turbine efficiency can be increased as well as the combustor outlet temperature.

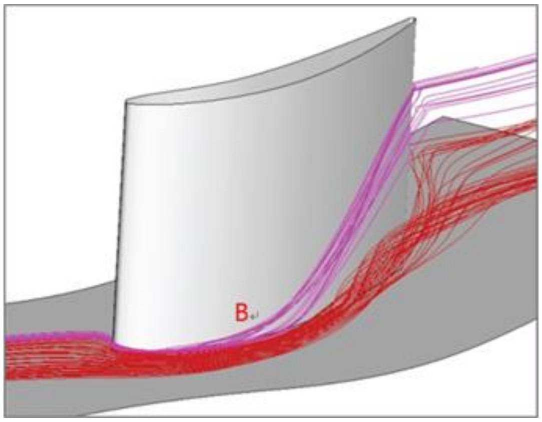

Figure 2
Secondary Flow Image from a CFD Analysis

Problem Approach using Fillets

Several methods like leading edge fillets and endwall modifications have been studied to weakening these vortices [2]. The fillets are added to the leading edge of the blade to smooth the transition to the endwall and are believed to reduce the adverse pressure gradient at the leading edge which creates the horseshoe vortex [2].

A CFD study will be perform using four different fillets to determine their influences over the secondary flows. The GE-E3 first stage turbine blade will be used since this has been the subject for previous studies of secondary flows.

Fillet 1 & 2 are large area fillets compare to fillets 3 & 4, covering more area on the suction side of the blade as seen in Figure 3. Fillets 2 & 3 have a much smaller height. The specifications for the fillets geometries are shown in table 1 providing the fillet height and the curves lengths for the OD curve and the ID curve.

Table 1
Fillets Specifications

		OD Curve		ID Curve	
	(Y/Smax) Height	Ss Length (in)	Ps Length (in)	Ss Length (in)	Ps Length (in)
Fillet1	7.3%	0.6172	0.3543	0.7771	0.5399
Fillet2	3.6%	0.5876	0.3296	0.7467	0.5176
Fillet3	3.3%	0.2830	0.2641	0.3239	0.4530
Fillet4	6.6%	0.3268	0.2900	0.3513	0.4741

Geometry and Mesh

For the study Fluent 15 and Ansys® meshing will be used to create the meshes and run the solution. The turbulence model selected was STT k-ω [3]. In order for the mesh to comply with the turbulence models requirement and obtain valid results, the Y+ must be between 3-10 which requires a fine mesh.

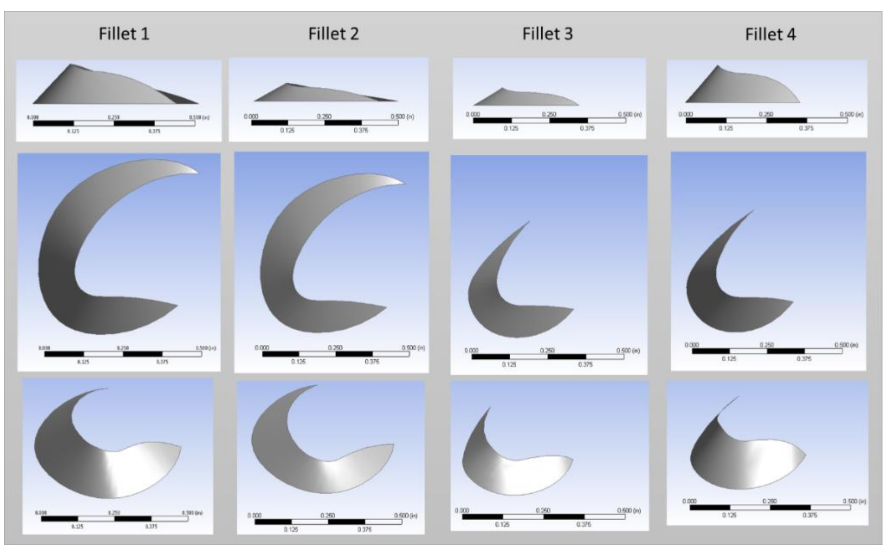

Figure 3
Fillets Geometries

To reduce the number of elements the original span of 1.6in. was reduce to 1in. since the area of focus is the bottom half of the geometry. The mesh was refined to half the volume size in the blade surface as seen in Figure 4.

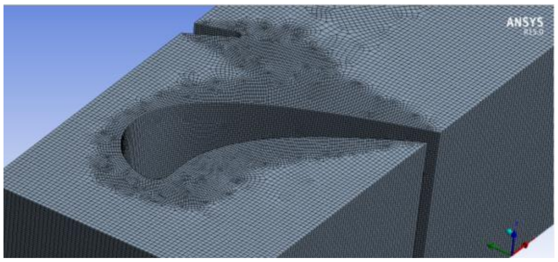

Figure 4
Refinement in Blade Surface

Only one sector of 5deg. was modeled using periodic. For simplicity purpose the geometry was cut straight forward as seen in Figure 5. Since periodic are been applied the fluid behavior will be the same as having several blades side to side. To achieve the required Y+ in the blade surfaces a Hexa mesh of 1.75 millions of elements was created.

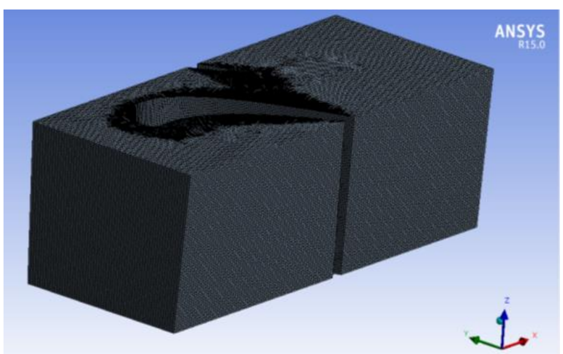

Figure 5
Mesh

Boundary Conditions and Case Setup

As previously mention for the case setup the turbulence model used was STT k-ω. For the inlet boundary conditions an Inlet Temperature of 25C and a Mach number of 0.3 were assume. The inlet and outlet pressure were stablish to be 101,506 Pa and 70,090Pa respectively [4]. The inlet flow angle was setup to be 41deg. [5]. The turbulence intensity and the length scale were 1.5% and 0.05in [1] [6]. For convergence criteria the cases were assume to be converge when the residuals achieve a steady state.

Five cases were study, one for each fillet and one without fillet as a baseline for comparison.

RESULTS

To visualize the results plots of pressure, velocity and turbulence kinetic energy were created. The first plots presented are the pressure plots at the endwall. For the pressure plots there is no significant difference between the case with no fillets and the cases with fillets as seen in Figure 6. One slight difference can be seen at the suction side of the case fillet1 and case fillet2 where the pressure is lower than the other cases.

Following the pressure plots are the velocity magnitudes pathlines created at the endwall in Figure 7. In these plots it is shown that the case without fillet has a very high velocity at the trailing edge compare to the rest of the cases. Case fillet2 has also a slight high velocity at the trailing edge. Cases with fillet 1 and fillet4 show and increase at the passage with high velocities at the pressure side of the blade. Since cases with fillet 1 and 2 are the cases with tall fillets, the high velocities might be related to the height of the fillets since the no fillet case and the shorter fillets cases doesn't show this behavior.

The plots of turbulence kinetic energy created at the trailing edge of the blade show that all the cases have a spot of high turbulence at almost the same place. Case with fillet4 show an increase of the turbulence compare to the baseline with no fillet. Cases with fillet 1 and 2 seem to have lower the turbulence in the spot. For fillet2 it seems like the turbulence might move to other areas since small spots were created at the trailing edge.

The last sets of plots are velocity magnitude pathlines created at the suction side of the blade where the vortexes are seen more clearly. All the cases seem to have large separation of flow since pathlines can be observe to be moving away from the endwall. Since all the plots are scaled the same, orange lines were drawn on the left side were the separation is evident.

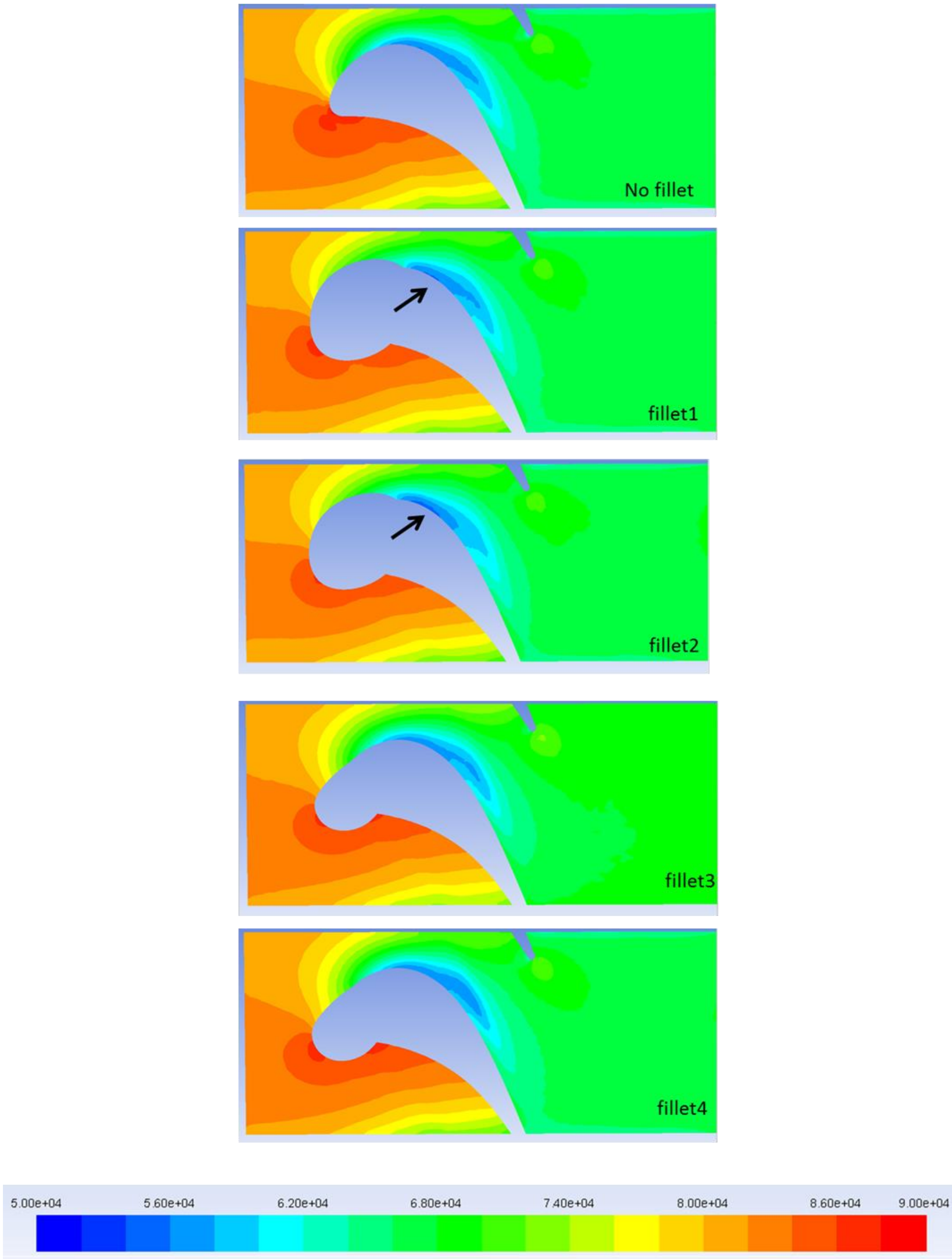

Figure 6
Static Pressure Plots of the Endwall in Units of Pascal

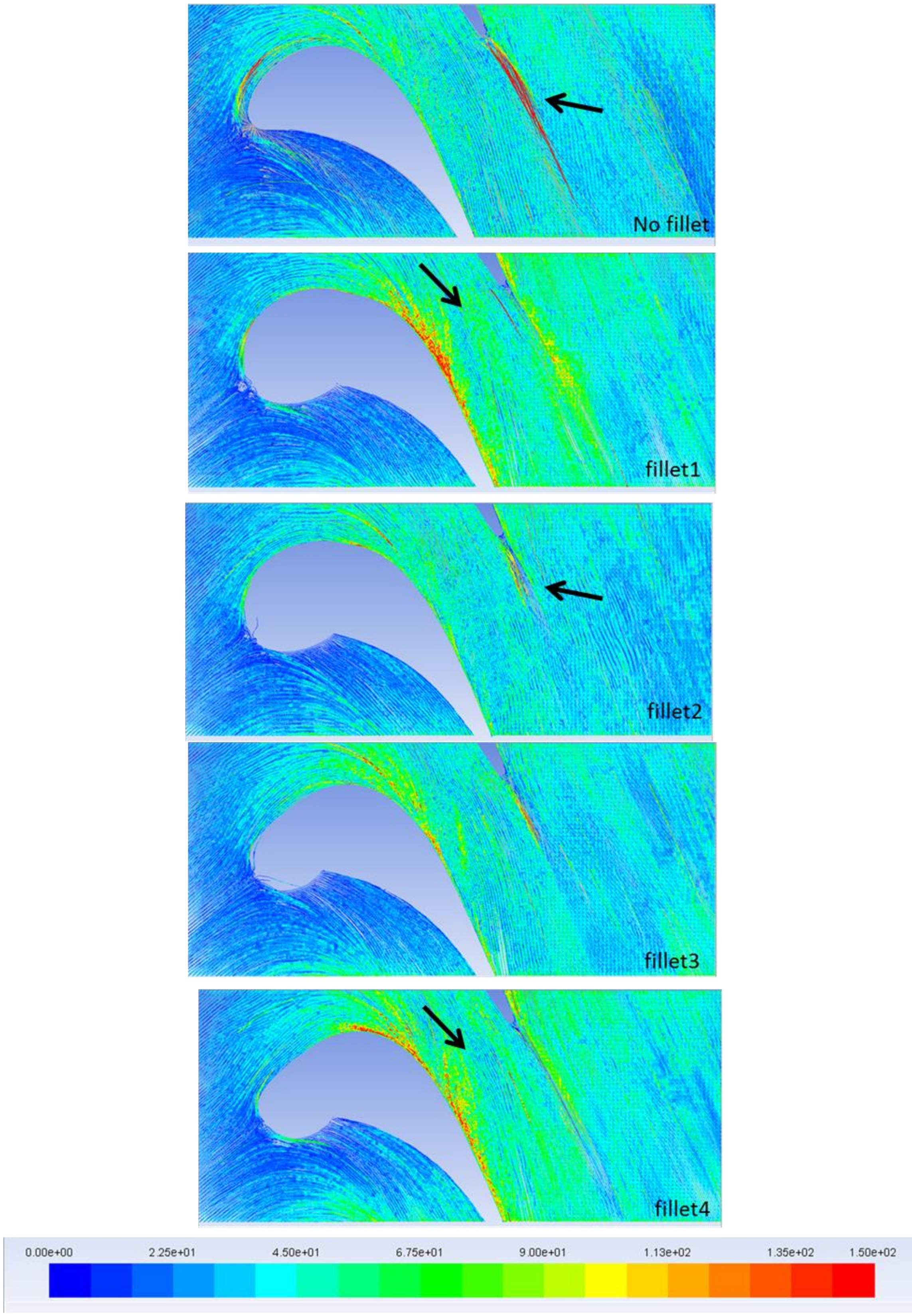

Figure 7
Velocity Magnitude Pathlines in the Endwall in Units of Meters per Seconds

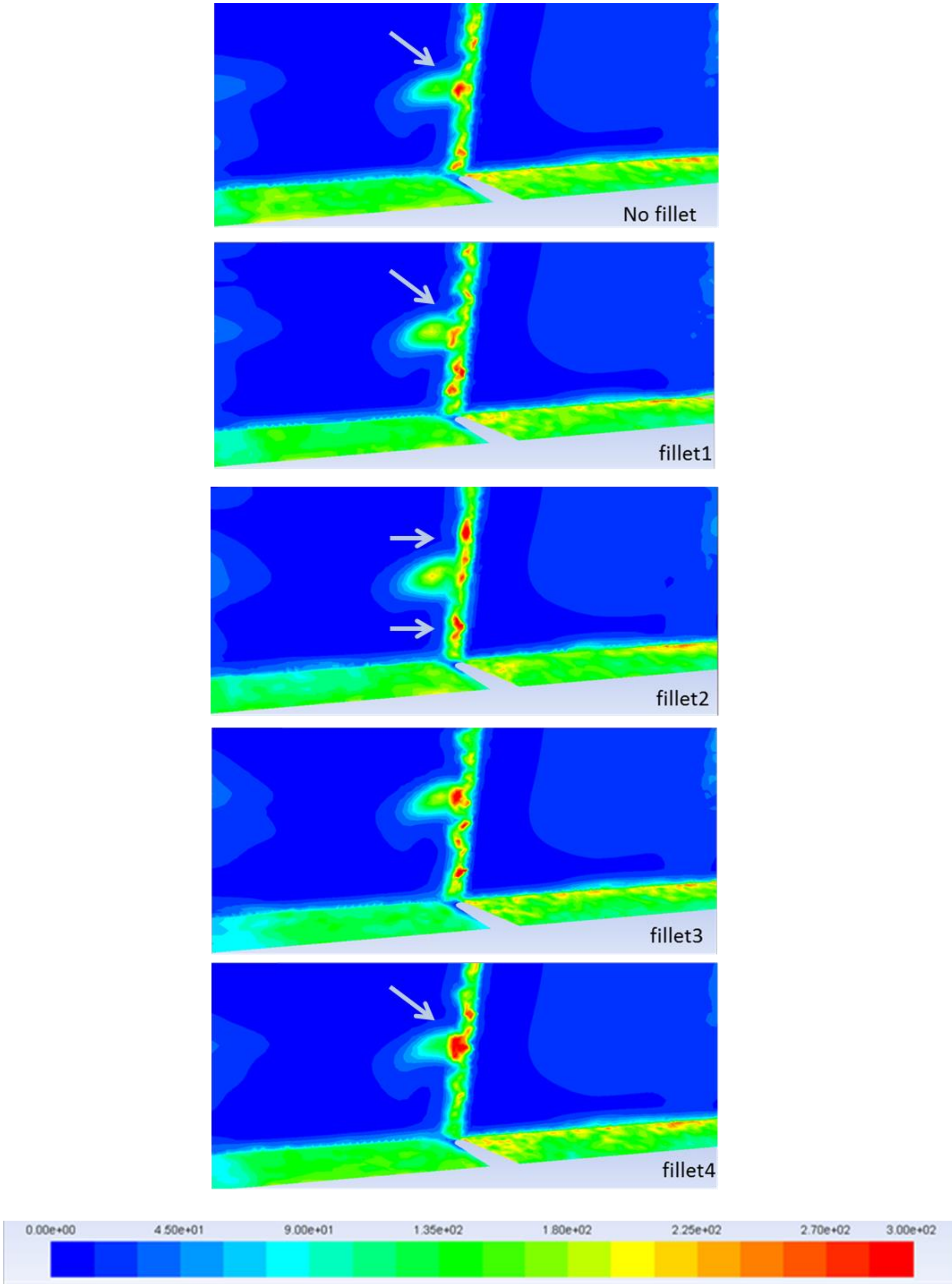

Figure 8
Turbulence Kinetic Energy Plot at the Trailing Edge in Units of Kelvin-Meter^2 per Seconds^2

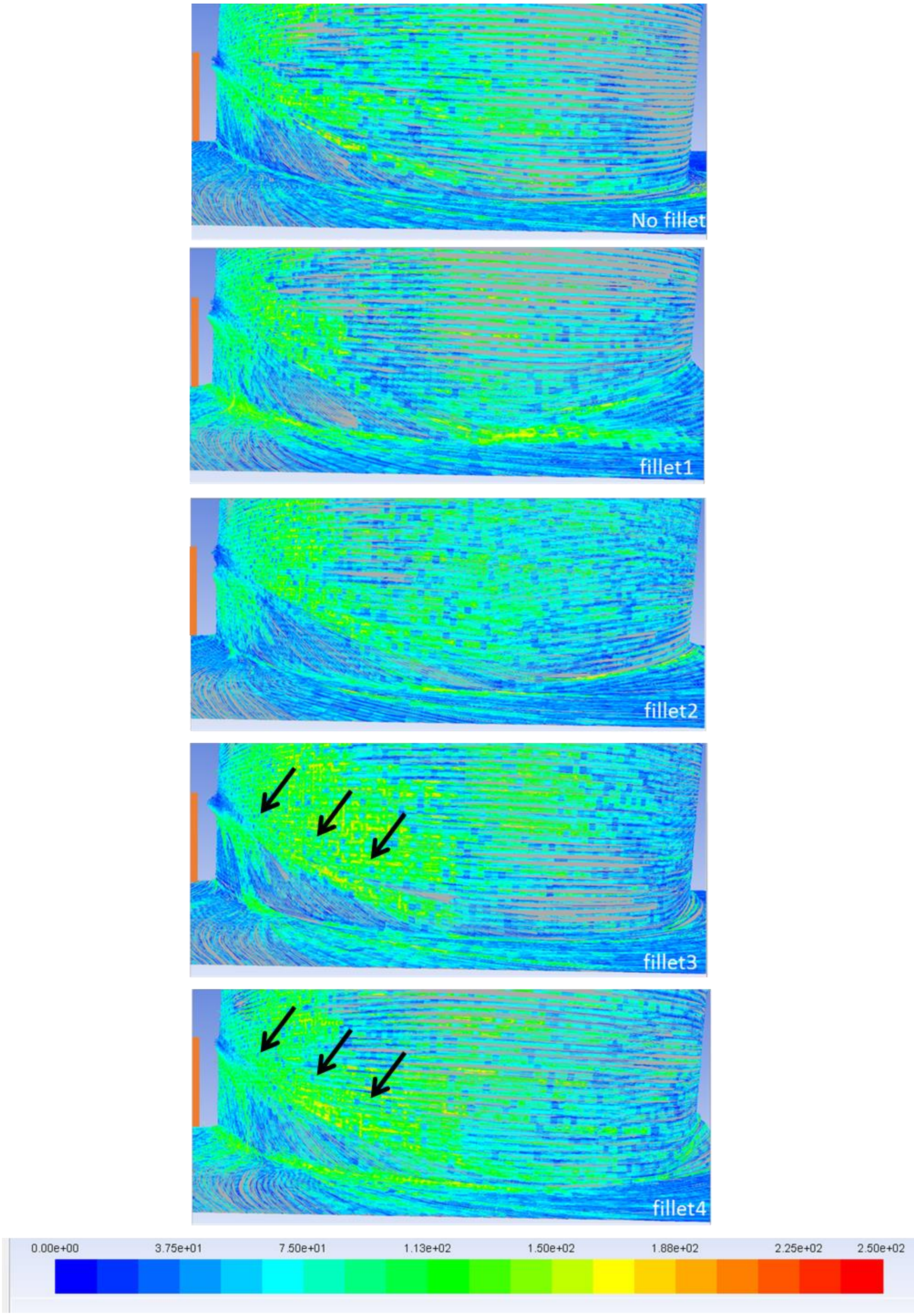

Figure 9
Velocity Magnitude Plot at the Pressure Side of the Blade in Units of Meters per Seconds

Comparing the separation distance from the endwall using the orange lines it can be seen that cases with fillet 3 and 4 with small area fillets show a small increase in flow separation distance.

CONCLUSION

The design fillets propose on this study didn't eliminate the vortex of the secondary flow. Although the vortex were not eliminated some effects were seen from the fillets. As it was shown in the velocity plots the tall fillets were able to increase the velocity in the passage which is a positive sign. For the other part the fillets with small area tend to worse the behavior by increasing the flow separation distance and in the case of fillet4 increasing also the turbulence kinetic energy at the trailing edge.

For future work it will be recommended to study other types of fillets to decrease the flow separation and as mention before it will also be good to explore the endwall modifications.

REFERENCES

[1] L. Yangwei, et al., "Numerical study of the effect of secondary vortex on three-dimensional corner separation in a compressor cascade" *Int. Journal of Turbo & Jet-Engines*, ISSN[2191-0332], DOI: [10.1515/tjj-2014-0039], February 2015.

[2] R. Gustafzon, "Flow and temperature measurements in a linear turbine blade passage with leading edge and endwall contouring with and without film cooling" M.S. thesis, Dept. Mech. Eng., Louisiana State Univ., May 2002.

[3] A. Andreini, et al., "Conjugate heat transfer calculations on a GT rotor blade for industrial applications Part II: Improvement of external flow modeling" ASME TURBO EXPO: Power for Land, Sea & Air, Copenhagen, Denmark, June 2013.

[4] T. Casey, et al., "Numerical and experimental study of film cooling effectiveness and total pressure loss for a realistic cascade with an annular endwall" 49^{th} AIAA Aerospace Science Meeting including the New Horizon Forum and Aerospace Exposition, AIAA2011-631, Orlando, FL, January 2011.

[5] J. Kullberg, "All experimental and numerical study of secondary flows and film cooling effectiveness in a transonic cascade" M.S. thesis, Dept. Mech. Eng., Univ. of Central Florida, Orlando, FL, 2011.

[6] V. Goriatchev, et al., "CFD- analysis of secondary flows and pressure losses in a nasa transonic turbine cascade" Conference on Modelling Fluid Flow, Budapest, Hungary, September 2003.