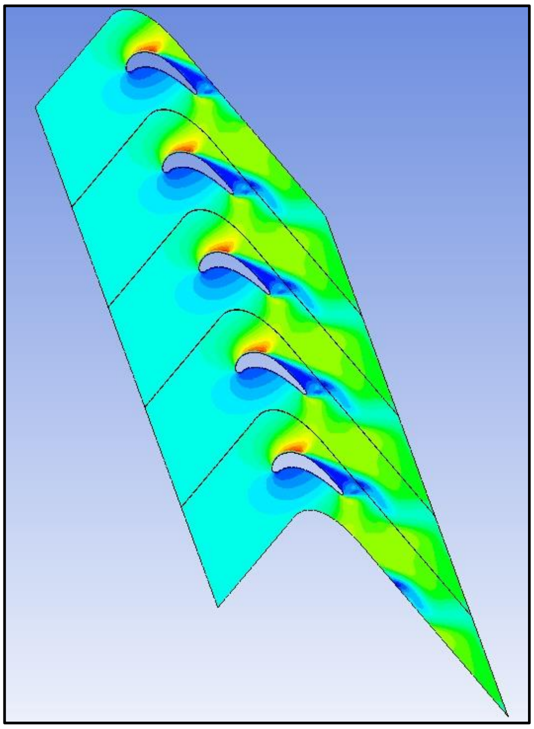
Turbomachinery and Propulsion (MECH 6171)

Project:

"CFD Analysis of the VKI Turbine Blade Cascade using Ansys CFX"



Submitted To:

Dr. Carole El Ayoubi

Submitted By:

Student Name:	Student ID:
Eshan Nigudkar	40166984

CONCORDIA UNIVERSITY
DEPARTMENT OF MECHANICAL, INDUSTRIAL & AEROSPACE ENGINEERING

Contents

List of Figures:

List of Tables

Introduction:

The development of modern compressor blade requires tremendous amount of study of airfoils and flow around it. The understanding of three-dimensional flow and losses across airfoils plays a significant role to achieve high efficiency and improved performance.

Computational fluid dynamics determines the fluid flow, heat transfer, mass transfer and other related parameters by solving a set of mathematical models using numerical methods. These tools are effectively used in industry and research and their robust nature supports the analysis of the fluid flow and performance of complex models such as compressors accessible.

In this project, the linear cascade of the VKI compressor blade is studied using the computational fluid dynamics method to obtain performance parameters to analyse the flow over the blade. The 3D CAD of the VKI blade was generated using the data points in SPACECLAIM. The model was then tessellated with a coarse mesh. Furthermore, a sensitivity analysis is done to get the grid independence. The suitable boundary conditions were assigned at the inlet, outlet, on the blade, casing, and hub to obtain numerically calculated performance parameters. These performance parameters include isentropic Mach numbers at the inlet, outlet, and blade and total pressure loss across the blade along with pressure, temperature, and velocity contours.

The 2D profile of the VKI compressor blade and flow path is given below:

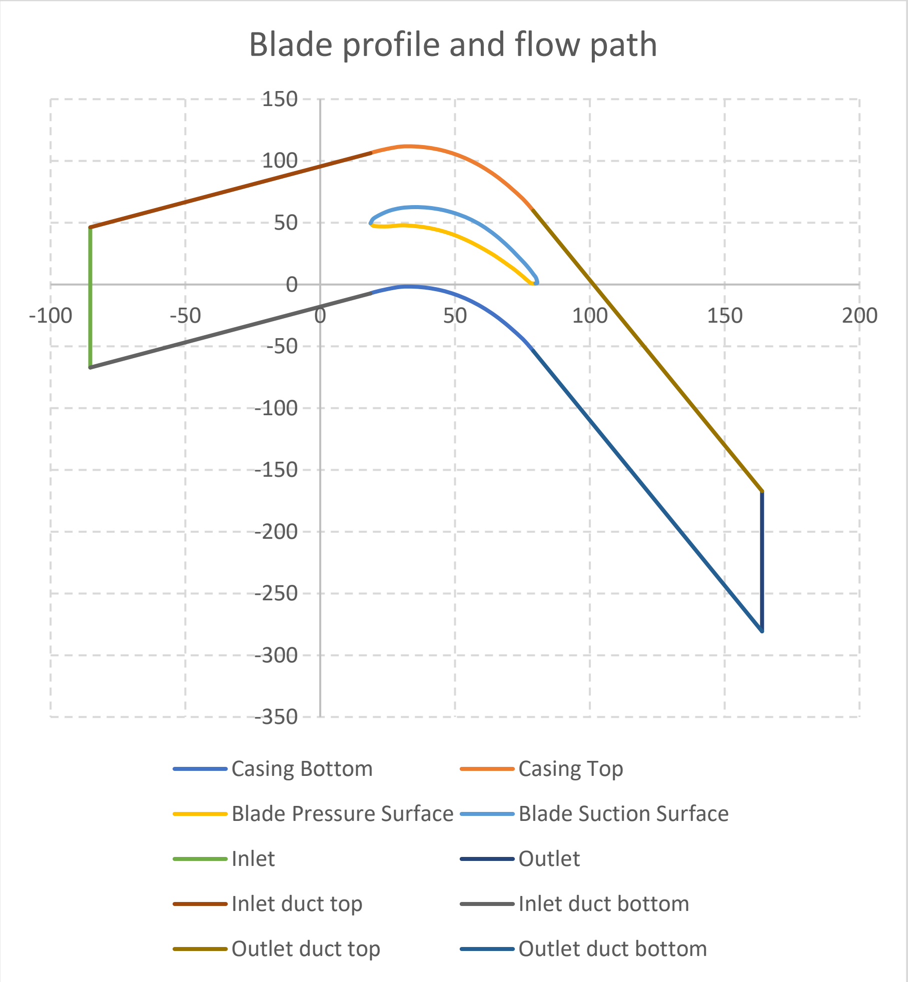


Figure 1: Airfoil profile and flow path.

Literature review:

Conservation of mass [1]

The principle of conservation of mass is that "mass cannot be created nor destroyed."
In a control volume this can be defined by mass coming in is equal to mass flowing out, when written in the equation form, it is,

$$\frac{d}{dt}\int_{V(t)} \rho(x,t)dV = \int_{v(t)} \sigma(x,t)dV$$

As we have a monophase flow, σ(x; t) = 0 and after deriving the above equation we

get the continuity equation.

$$\frac{d\rho}{dt} + \nabla\vec{V} = 0$$

For a compressible steady flow study, the divergence of the density and velocity product is equal to 0, that integrated on the volume implies that the mass flow rate is conserved.

Conservation of momentum [1]

The principle of the conservation of momentum or Newton second law of motion states that, "The overall force over a system is defined by the change of the momentum of the system ". Below is given in the X direction momentum change relation:

$$\rho(\frac{Du}{Dt}) = \rho X_x - \frac{dp}{dx} + \mu(\frac{d^2u}{dx^2} + \frac{d^2u}{dy^2} + \frac{d^2u}{dz^2})$$

Total derivative = (Pressure gradient + Body force term + Diffusion) in the x direction.

Similarly, can be determined for y and z directions.

Conservation of energy [1]

Conservation of energy or first law of thermodynamics state that the total energy of a system is conserved. This is expressed by the rate of energy in a volume equates to the heat Q and Work W transfer in the same volume. In equation form it can be written as:

$$\frac{d}{dt}\int_{V((t)} EdV = W + Q$$

By solving the above equation in X direction, we obtain:

$$\rho\frac{De}{Dt} = -p\frac{\partial u_i}{\partial x_j} + \mu\frac{\partial u_i}{\partial x_j}(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_i}{\partial x_j} - \frac{2}{3}\frac{\partial u_r}{\partial x_r}\delta_{ij}) + \frac{\partial}{\partial x_i}(\frac{\partial T}{\partial x_i})$$

With u corresponds to the velocity in X direction T corresponds to the Temperature.

Turbulence model k-ε[1]

In general, the k-ε model is a turbulence model that supports the simulation and prediction of the flow for turbulent environments away from the boundary walls. The turbulence of the kinetic energy and turbulence of the dissipation terms are considered. The model is suitable when there are high pressure gradients, to study turbulent behaviour away from boundary walls and complex geometries. The model is not recommended when boundary layer flow near the wall is studied. There is a difference in the k-ε model of the CFX and fluent solver, it is given below

1) For turbulent kinetic energy:

$$\frac{\partial(\mathrm{k})}{\partial \mathrm{t}}+\frac{\partial}{\partial \mathrm{x_j}}(\mathrm{U_j k}) \;=\frac{\partial}{\partial \mathrm{x_j}}[(\mu+\frac{\mu_t}{\sigma_k})\frac{\partial \mathrm{k}}{\partial \mathrm{x_j}}] \;+\mathrm{P}\,-\,\rho\epsilon+\,\mathrm{P_{kb}}$$

2) For turbulence dissipation:

$$\frac{\partial(\epsilon)}{\partial \mathrm{t}}+\frac{\partial}{\partial \mathrm{x_j}}(\mathrm{U_j}\epsilon) \;=\frac{\partial}{\partial \mathrm{x_j}}[(\mu+\frac{\mu_t}{\sigma_\epsilon})\frac{\partial \epsilon}{\partial \mathrm{x_j}}] \;+\;\frac{\epsilon}{\mathrm{k}}\;\;(\mathrm{C_{\epsilon 1}P_k}\,-\,\mathrm{C_{\epsilon 2}}\epsilon+\,\mathrm{C_\mu P_{\epsilon b}}\;)$$

P_{kb} & $P_{\varepsilon b}$: effect of buoyant forces; P_k : turbulence generated by viscous forces. Constants $C_{\varepsilon 1}$ = 1:44 ; $C_{\varepsilon 2}$= 1:92 ; $C_{\varepsilon 3}$= 0:09 ; σ_ε = 1:3 ; σ_k= 1.

From the Research community:

Cascade models of aerofoils has been a topic of research and investigations among the researchers. They are crucial for the development of the highly efficient and better performing turbomachines.

One of the thorough research conducted by the von Karman institute for fluid dynamics.[2], The set of research papers by name 'Flow in compressors' explains the complete flow behaviour of the compressors. One of the papers, suggests that to speed up the process of an aero-mechanical optimization by the application of Computational Fluid Dynamics (CFD) solvers on Graphics Processing Units (GPU). A study on the reveals that the turbines operating in the transonic regime experience shock waves. To limit the mechanical stress induced by the vane-rotor shock interaction, it is proposed in the second part of the paper to optimize the vane geometry to mitigate its downstream distortion.

S. Pittala et al. [3] conducted CFD analysis on a linear blade cascade to analyse the performance parameters such as pressure, temperature, velocity. The flow generation in the analysis was applied to support the design process of turbine blades at different flow conditions. The CFD simulation of VKI blade in this project bear a resemblance to many aspects of the work carried by S. Pittala.

A numerical simulation investigation on three-dimensional turbulent flow in a transonic turbine cascade by V.D. Goriatchev et al. [4] were conducted on two CFD codes and investigated with measurements of a linear cascade at the NASA GRC transonic turbine blade cascade facility. The simulation provides a thorough understanding into the characteristics of secondary flow development in cascade of turbine blade with high turning angles.

Problem Statement:

In this project, the study of a VKI turbine blade is done over a control volume to analyse the flow pattern and performance parameters that can be used further to improve efficiency and performance. The analysis will be done using the CFD tools. One of the main goals of this project is to become acquainted with one of the numerous commercially available CFD software, CFX.

The VKI turbine blade is a typical 1st stage gas turbine blade. The cylindrical design of the blade has same 2D cross section from hub to tip. The objective of this project is to do model set-up and simulate a linear cascade of VKI blade using Ansys CFX tool, which will be further post processed to obtain typical turbine performance parameters.

Following parameters will be obtained to assess the performance of the blade.

1) Isentropic Mach number distribution at the inlet and exit of the of the cascade.
2) Plot the isentropic Mach number distribution over the suction and pressure surface of the blade.
3) Total pressure loss across the blade.
4) Contours of the velocity, pressure, and temperature across the cascade.

Methodology:

In this project following steps will be performed to achieve the performance parameters stated above.

Geometry Preparation:

1. Generating the 3D geometry of the VKI turbine blade from the data points of the blade profile and flow path geometries in the SPACECLAIM, a CAD handling software of Ansys.

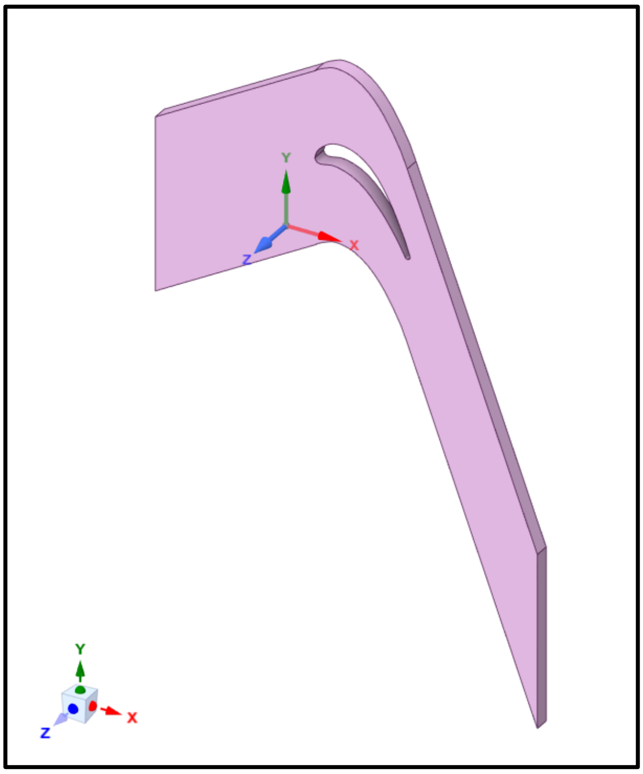


Figure 2: Isometric view of VKI turbine blade.

Table 1: Blade Parameter.

BLADE PARAMETER	VALUES
True Chord Length	80mm
Span	10mm to 20mm
Stagger angle	38.5°
Inlet blade angle	30°
Inlet flow angle	30°
Outlet flow angle	69.5°
Pitch	113.5mm

Tessellation:

The above CAD then will be tessellated using Ansys workbench meshing applet, using appropriate meshing technique given below. A mesh sensitivity analysis will be performed to achieve grid independency and results on final mesh count will be discussed in the next section.

Table 2: Mesh technique and parameters.

Mesh Type:	Hex Dominant (quadratic)
Number of divisions in edge sizing	80
Mesh element size:	8 mm – 3.2 mm
Curvature capture	Yes
Inflation Layer	First layer thickness of 0.1mm for 10 layers.

Using a coarser mesh of 8mm at first, a CFD simulation will be performed using Ansys CFX software where following boundary condition will be supplied to the tessellated geometry. The Hex dominant quad mesh is chosen because it takes less time for meshing and gives optimal results when compared with tetrahedrons where mesh element counts increase significantly for a similar element size. Due to less nodes, the numerical simulation also utilizes less memory space and time to complete analysis. The number of divisions of 80 and inflation layer around the blade helps in the perfect capture of the blade geometry. Finally, curvature capture also helps in optimal capture of the curvatures over the complete geometry of cascade.

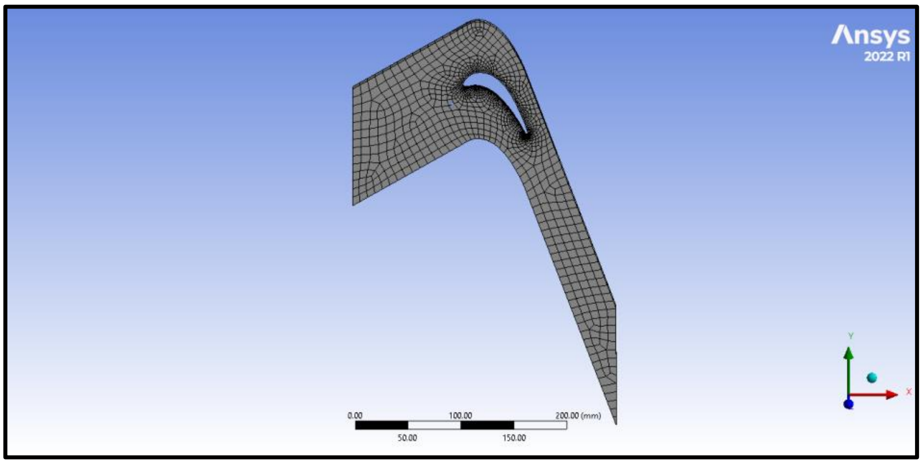


Figure 3: Base mesh (Coarse).

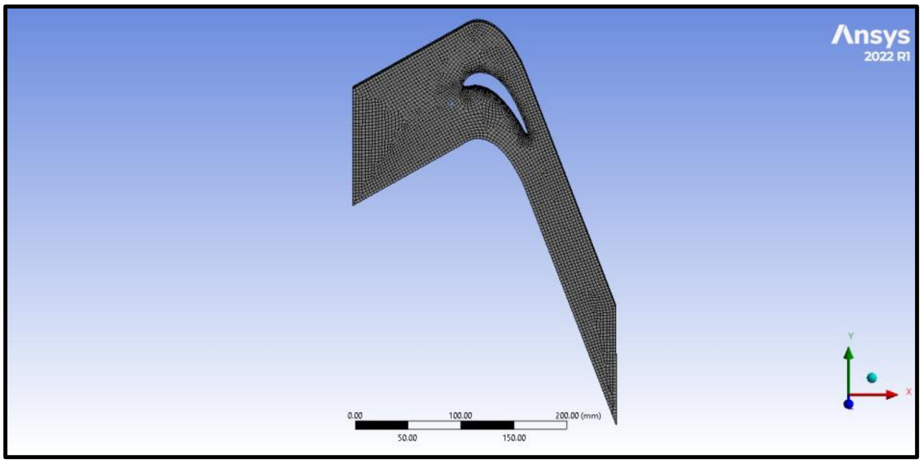


Figure 4: Optimized Mesh.

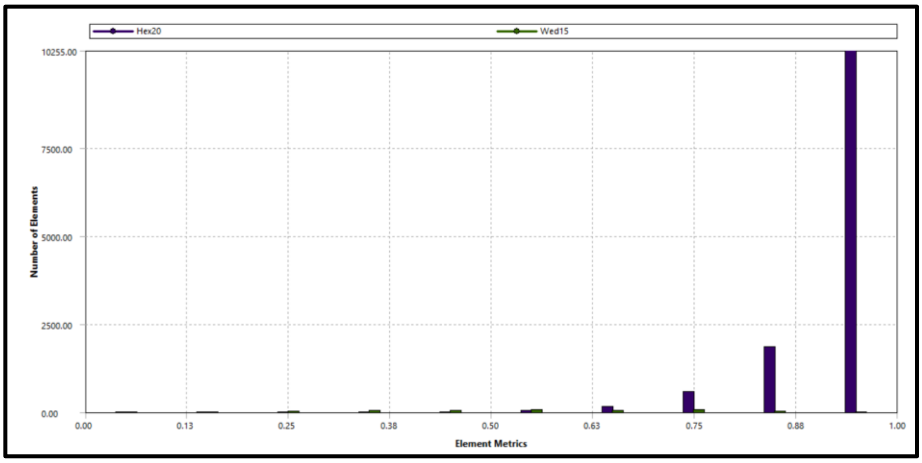


Figure 5: Element quality matrix.

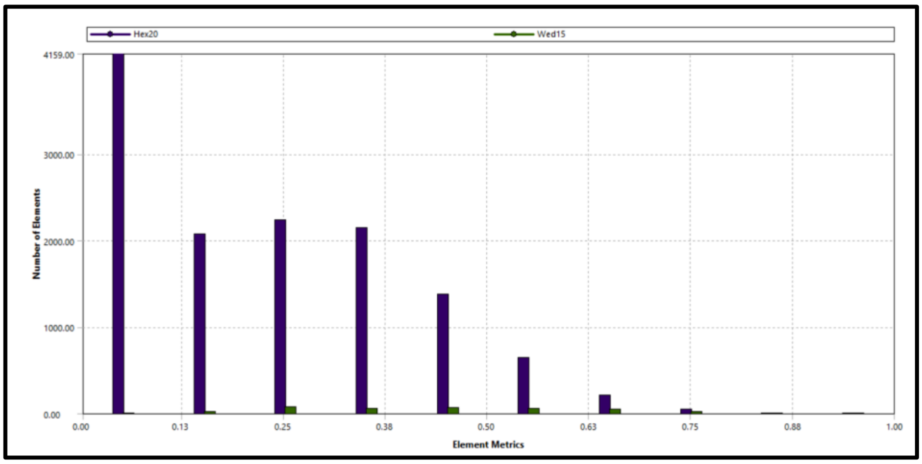


Figure 6: Skewness matrix.

Model Setup:

The meshed model generated is imported into CFX Pre for pre-processing. The model is set for steady state condition and boundary conditions were specified in the default domain. The k-ε model is considered because the scope of this study is to observe flow after the buff body. Translational periodicity was applied for blade, hub, and tip of the cascade.

The wall boundary conditions are specified as given below.

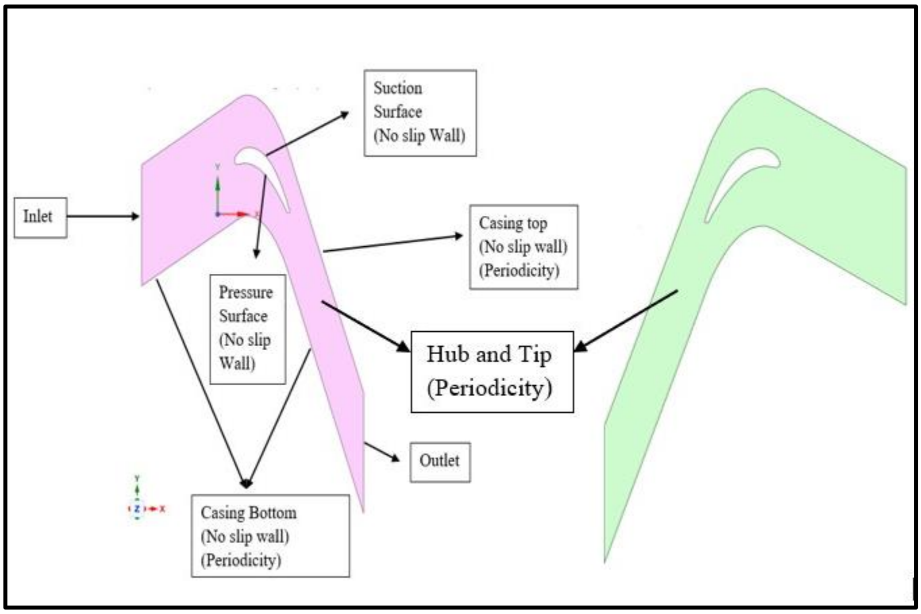


Figure 7: Wall boundary conditions of the geometry.

Table 3: Boundary Setup Used in CFX-Pre

PARAMETERS	CONDITION
Total Inlet Temperature	415 K
Total Inlet Pressure	288 kPa
Outlet Static Pressure	165.5 kPa
Turbulence Model	k-ε model
Turbulence Intensity	Medium
Blade wall temperature	290 K
Inlet flow angle	30^o
Energy model	On
Casing Top	Translational Periodicity
Casing Bottom	Translational Periodicity
Hub	Translational Periodicity
Tip	Translational Periodicity

Table 4:Conditions Used in Solver Control

PARAMETER	CONDITION
Advection Scheme	High Resolution
Turbulence Numeric	High Resolution
Maximum Iterations	2500
Timescale Control	Auto Timescale
Timescale Factor	0.1
Residual Type	RMS
Residual Target	10-6

Solution: CFX Solver:

In the CFX-Post, we assign the solution conditions as parallel processing using 6 cores with double precision. The solution is set to run for 2500 iterations or convergence with a timescale of 0.1. The simulation will be executed until a convergence is reached of absolute values and numerically.

Post processing using CFX-Post.

The results obtained from the simulation will be post processed in the CFX post applet of the Ansys workbench. The contours of the static pressure, total pressure, temperature, velocity, and isentropic Mach numbers are obtained over the domain. Several planes are created over the pressure and suction surface of the blade to get the isentropic Mach number distribution over the blade.
This process will be followed for finer mesh settings until a mesh independency is achieved. The post processing to get the performance parameters is carried out in the CFX-Post of the Ansys software. The result discussed below is of the finer mesh setting of around 70000 nodes.

Results and Discussion:

Convergence

Convergence is a way of understanding that CFD simulations are reaching towards a value where conservation equations such as momentum, continuity and energy are satisfied in all the cells to a given specified tolerance, in this case it is 10^{-6}. The plot lines of conservation equations seem to stabilize or follow a recurring pattern.

In this project, the simulation is performed with high resolution for advection and turbulence modelling. The simulation with nine different values of nodes for the sensitivity analysis is executed. Below plot depicts the RMS residuals of the finest mesh produced on the model (3.2 mm). The initial convergence is achieved by the base mesh of 8 mm and after that a finer mesh model is simulated with taking initial values from the first run.

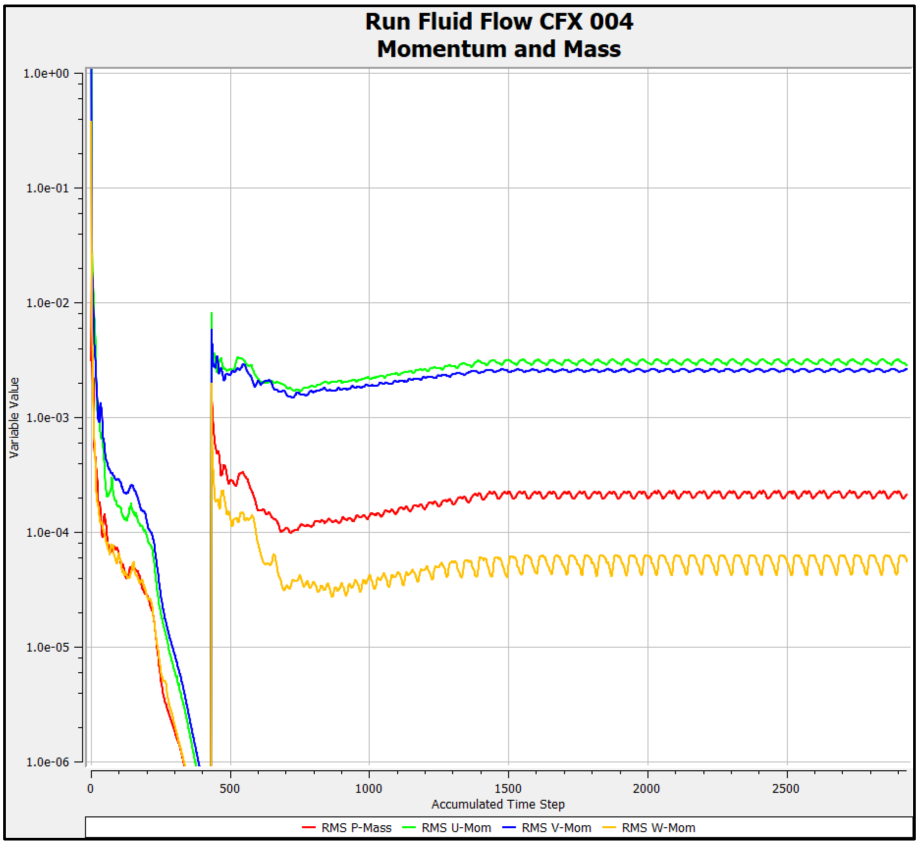


Figure 8: RMS convergence graph for Mass and Momentum.

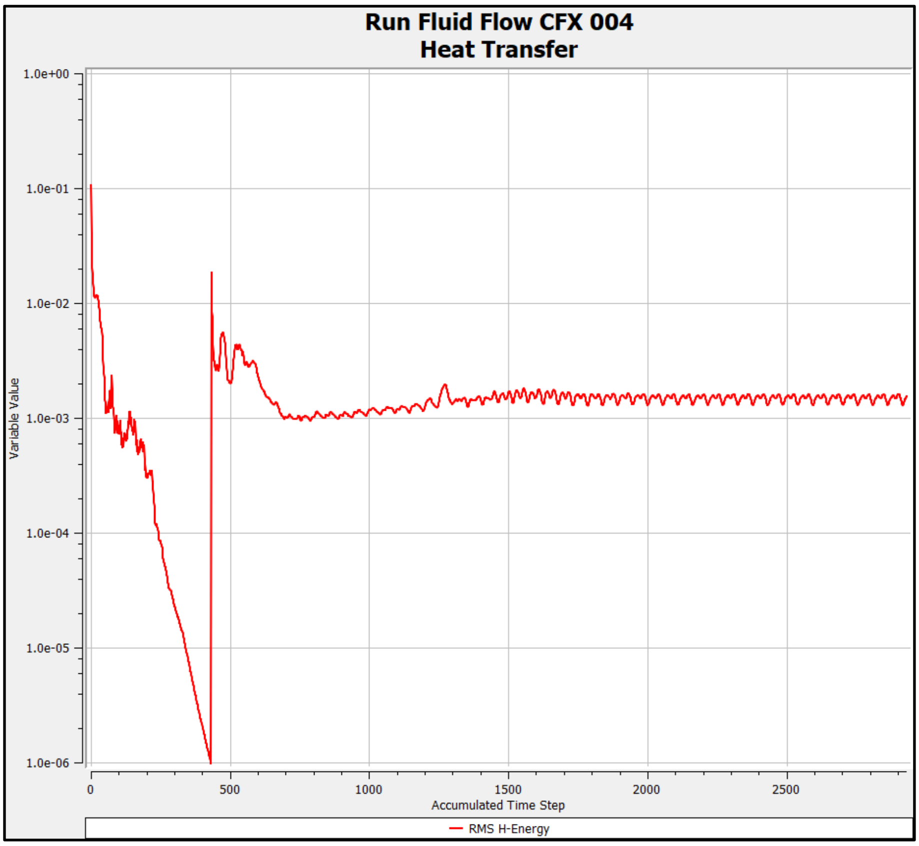


Figure 9:RMS convergence graph for heat transfer.

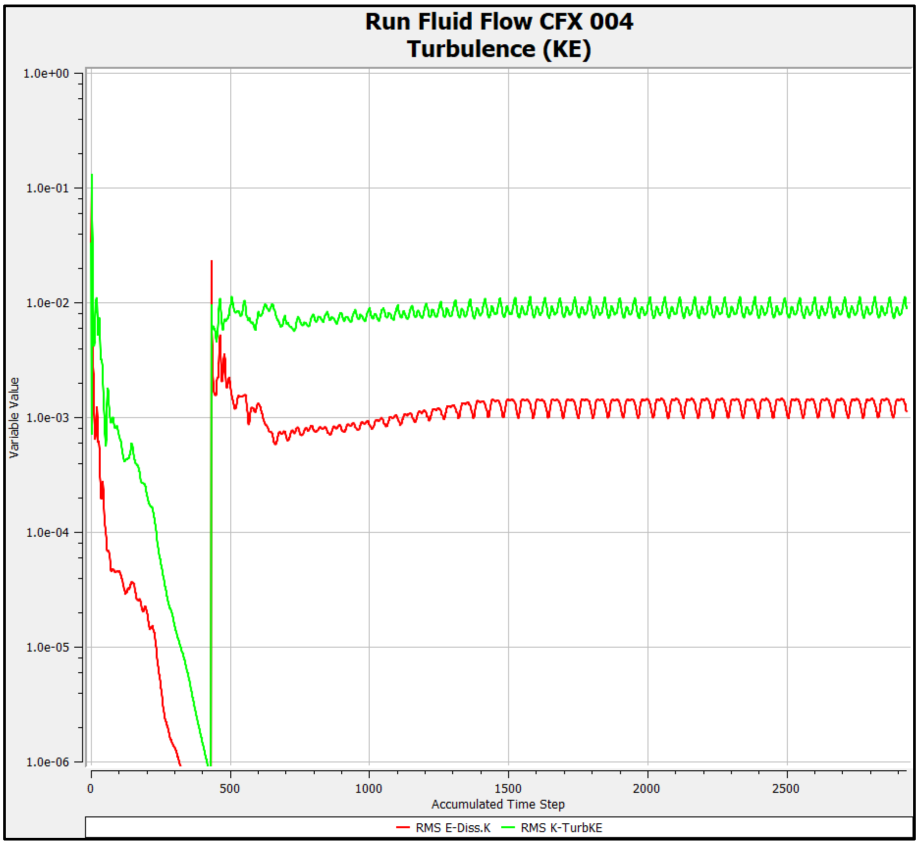


Figure 10:RMS convergence graph for turbulence.

The above plots justify the numerical convergence. For absolute convergence, a normal force distribution in the X direction on the pressure surface is plotted in CFX Post to confirm the absolute convergence. The change in the value reduces to a stable value after 1600 iterations.

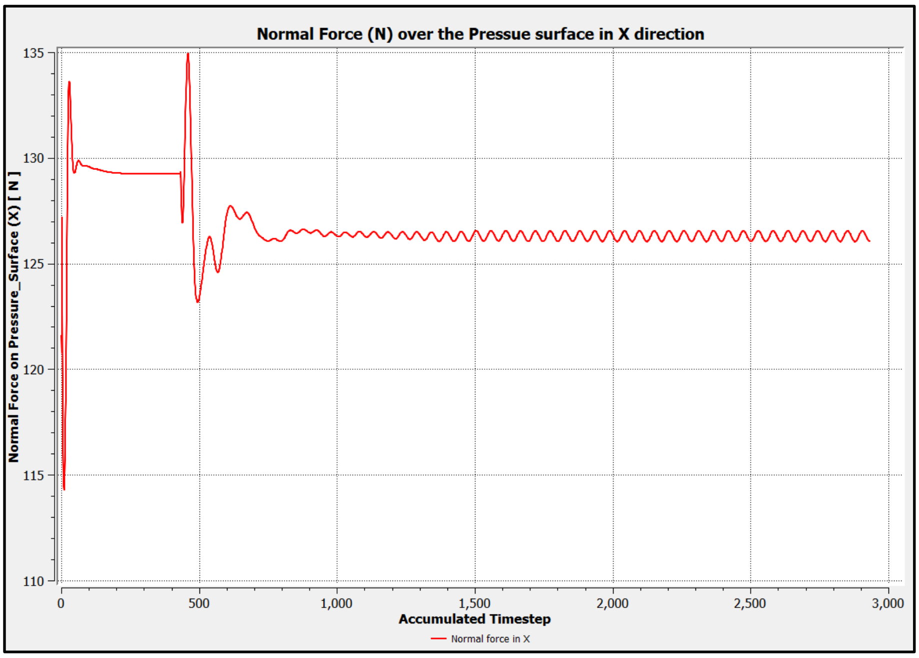


Figure 11:Absolute convergence displayed by normal force distribution.

Grid independence of the solution:

For 9 different mesh sizes, the isentropic Mach numbers at the inlet and outlet of the blade are compared to demonstrate the solution's mesh independence. The outcome is tabulated and graphically displayed below. The optimal mesh obtained is around 45,248 nodes as the results are similar to the order of 10^{-3}.

Table 5: Mass flow out and isentropic Mach numbers at inlet and outlet of the cascade to demonstrate mesh independency

S No.	Mesh Nodes	Mach at Inlet	Mach at outlet	Mass flow out [kg s^-1]
1	21972	0.379072	0.7437	0.337908
2	25480	0.377863	0.720746	0.33697
3	28100	0.380604	0.76617	0.339026
4	32033	0.379179	0.74261	0.33797
5	38758	0.377698	0.740852	0.336867
6	**45248**	**0.381435**	**0.76269**	**0.339639**
7	48562	0.38159	0.765609	0.339753
8	52814	0.380805	0.768934	0.339172
9	61366	0.380797	0.769638	0.339163

Graph Plot:

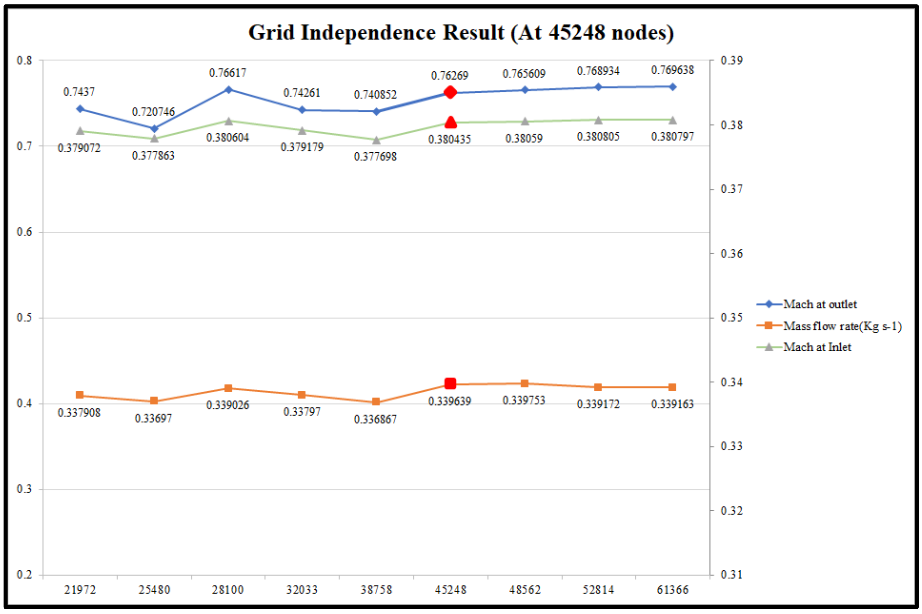


Figure 12:Graph demonstrating grid independency.

Isentropic Mach Number:

Isentropic Mach number is defined the Mach number assuming no losses in the flow. It is derived from the isentropic flow relations as given below.

$$\frac{pt}{p} = \left(1 + \gamma - \frac{1}{2} M^2\right)^{\frac{\gamma}{\gamma-1}}$$

Where,
pt is the total pressure in the in the free flow.
p is the local static pressure.
γ is the specific heat ratio and
M is the isentropic Mach number.
From the above equation, Isentropic Mach number is,

$$M = [(\frac{pt}{p})^{\frac{\gamma}{\gamma-1}} - 1] * 2/(\gamma - 1)$$

Isentropic Mach number is not pre-programmed calculation in the Result applet. Therefore, a new variable is created by making a function of the above equation in the expression tab. Thus, Mach numbers are calculated over the blade, at the inlet and the outlet.

Table 6: Chord ratio(x/c) vs Isentropic Mach number

x/c	Isentropic Mach Number Over Suction Surface	Isentropic Mach Number over Pressure Surface
0.23125	0.4923	0.4951
0.3125	0.8581	0.2099
0.375	0.8625	0.2031
0.4375	1.0368	0.2167
0.5	1.0942	0.2155
0.5625	1.1257	0.2220
0.625	1.2733	0.2678
0.6875	1.2358	0.2995
0.75	1.0244	0.3741
0.8125	0.8623	0.4573
0.875	0.7962	0.5756
0.9375	0.8011	0.6915
1	0.6227	0.6227

To calculate the isentropic Mach number over the blade, multiple planes are created over the blade and then the Mach number is calculated using the above formulation. The values obtained over the planes are average values of total pressure and static pressure from the CFX post and plotted below using Microsoft Excel.

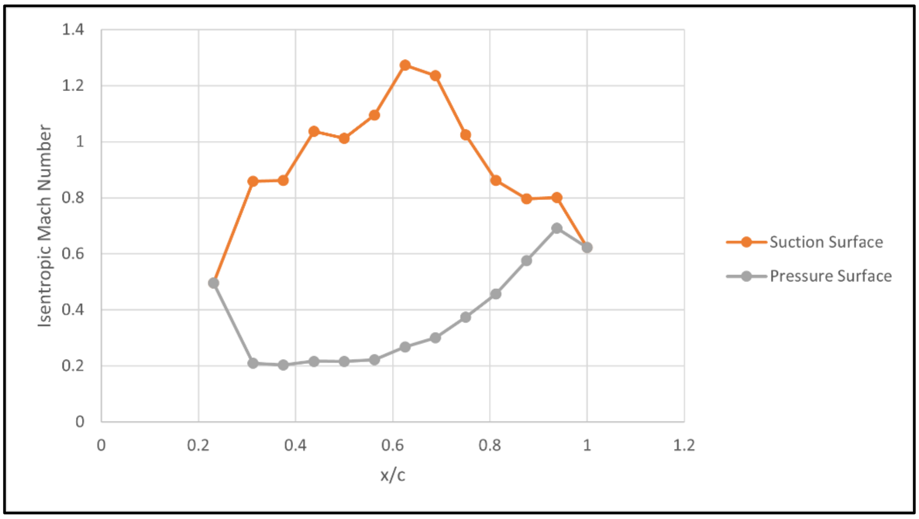


Figure 13:Graph displaying Isentropic Mach number vs x/c

Total Pressure and Static Pressure contours:

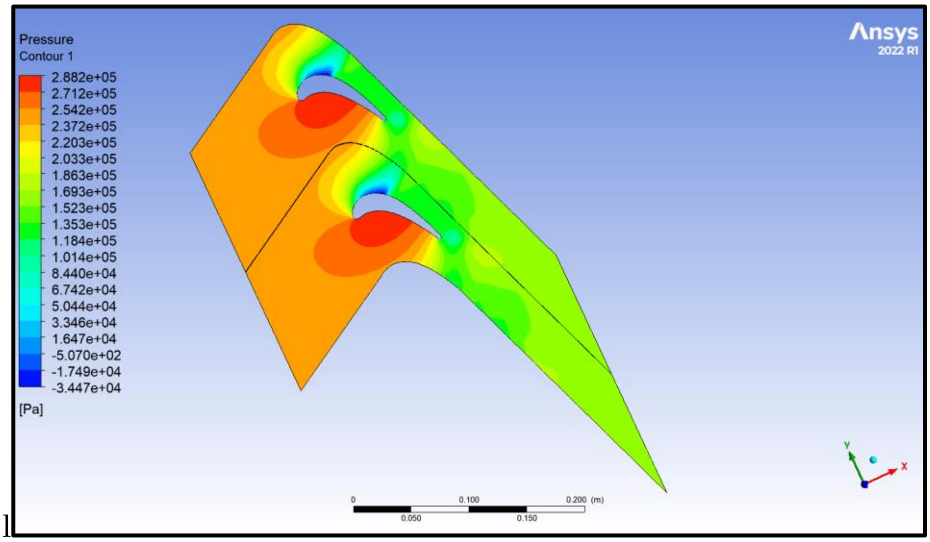


Figure 14:Static pressure contour over the Cascade.

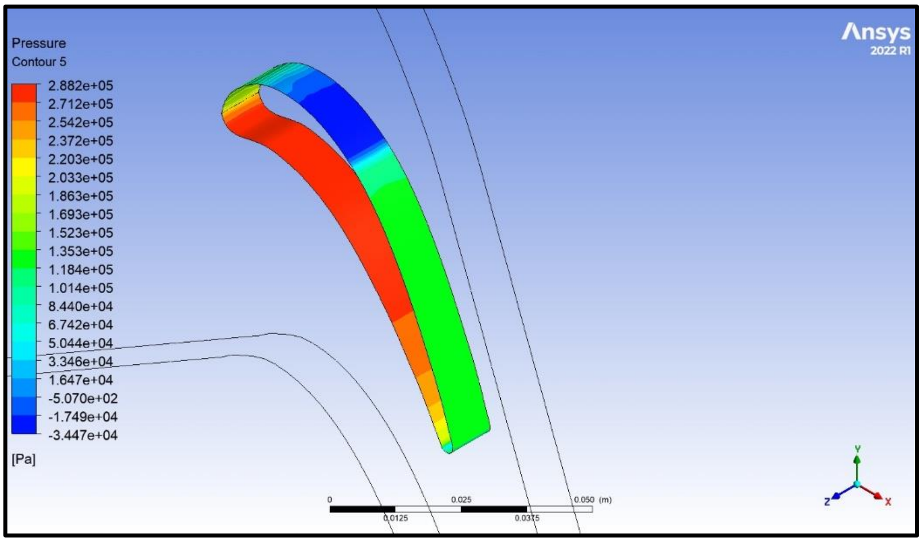


Figure 15:Static pressure contour over the Blade

The pressure contours of the cascade are shown above. However, to better study the flow characteristics, the focus will be on one contour and the other one is similar to it. From the figure 9, the pressure dips from the inlet to the outlet of the casing due to the aerodynamic blade angle. Certainly, once the flows touch the leading edge at certain angle, a pressure difference is established between the pressure surface and the suction surface of the blade as observed from figure 10 as well.

Looking at the blade, the pressure of the pressure surface is higher in comparison to suction surface. Thus, these result match the characteristics of the turbine blade that tends to accelerate the flow. Observed closely, the suction surface at one point turns to negative pressure that results in the flow separation later over the suction surface. This is due to a severe reduction of the pressure in comparison to the severe increase of velocity that can be explained as an adverse pressure gradient that led to the development of the separation flow.

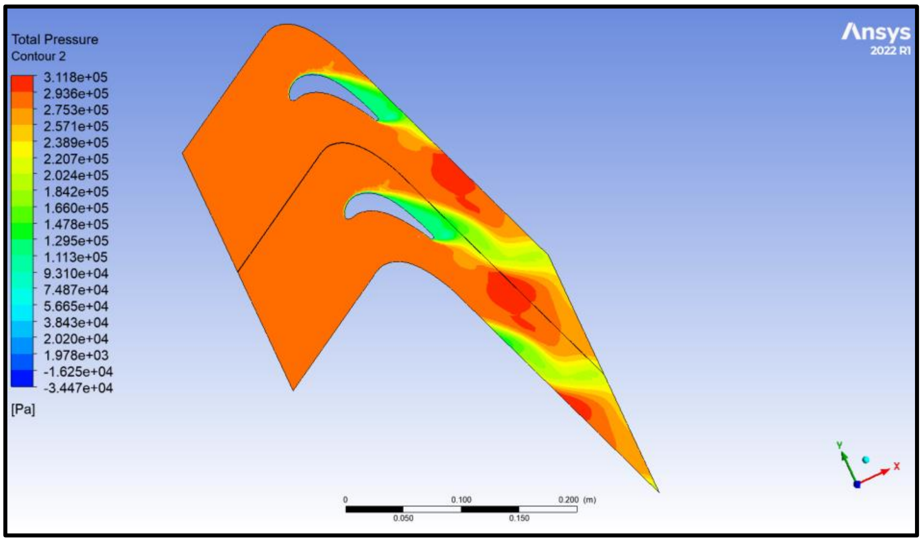


Figure 16:Total pressure contour over the Cascade

Looking at the total pressure, there is a huge pressure difference between the flow separation region and the flow region. However, in the static pressure contour, it sticks to a constant value in the flow separation region. Therefore, it can be inferred that the total pressure is a sum of the static pressure and the dynamic pressure, where the latter is related to the velocity from the mathematical formulation. As it can be seen from the velocity contour that the velocity is next to nothing, thus dynamic pressure value has analogous value. The total pressure grows to be close to the static pressure value and experiences a sudden drop. This high-pressure drop certainly relates to the overall pressure loss occurring due the separation flow.

Total Pressure loss across the blade:

The pressure loss across the blade is determined by creating two planes at both leading edge and trailing edge of the blade.

Total Pressure Loss is given by,

$$\Delta ptloss = \text{avg}\ (ptleading\_edge) - \text{avg}(pttrailing\_edge)$$

$$= 285.981 - 177.530$$

$$= \mathbf{108.451\ kPa}$$

Temperature:

The temperature across the cascade reduces as the flow goes over the blade from inlet to outlet. The maximum temperature is observed at the pressure surface of the blade. At inlet, the temperature recorded was 403.598K and at outlet it was found to be 398.349 K. The difference in the value from the boundary condition of inlet temperature is because the value in the contour is static temperature values, the supplied value is total temperature value at the inlet which is 415 K.

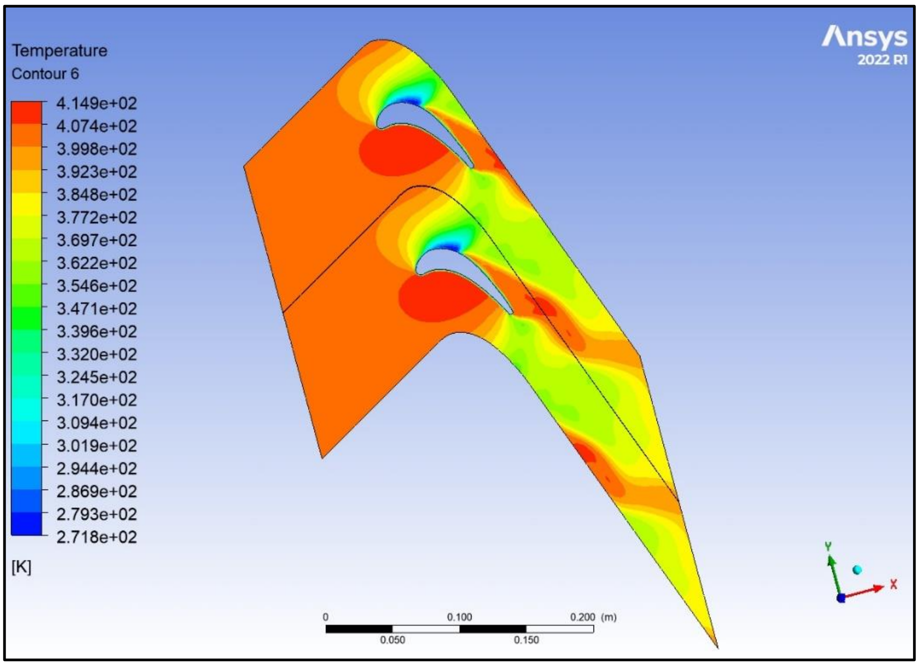


Figure 17:Temperature contour of the cascade.

Velocity:

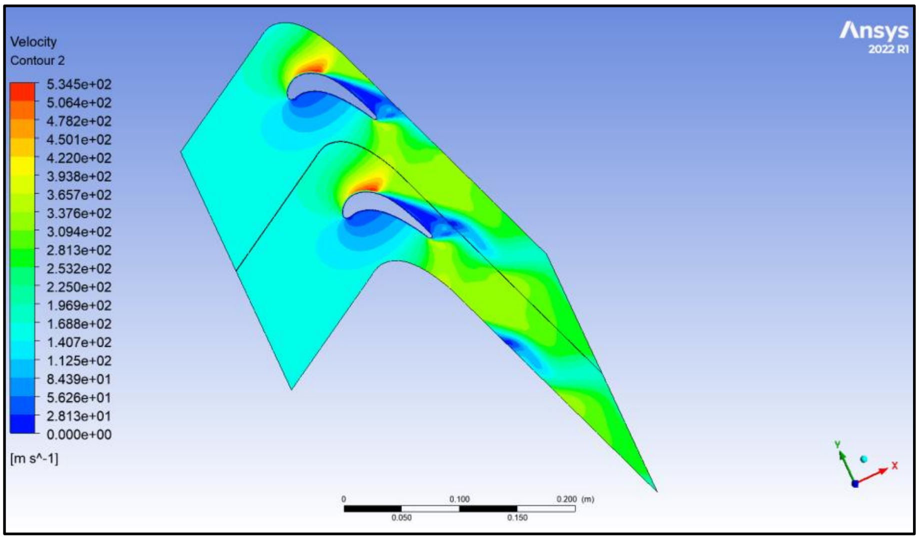


Figure 18: Velocity contour of the blade.

From the inlet to the leading edge of the blade, the velocity is almost constant. As it reaches the leading edge, it is subjected to acceleration over the suction surface, where velocity reaches to a maximum and therefore to a maximum isentropic Mach number right before the separation flow where the velocity is zero. A zero-velocity region is also observed on the casing near to exit, this is due to the assumption of translational periodicity on the top and bottom surface, an infinite cascade. Therefore, this null velocity at this region comes from the blade that is just under it. This is better represented in figure 18.
Below is a complete cascade for the velocity contour:

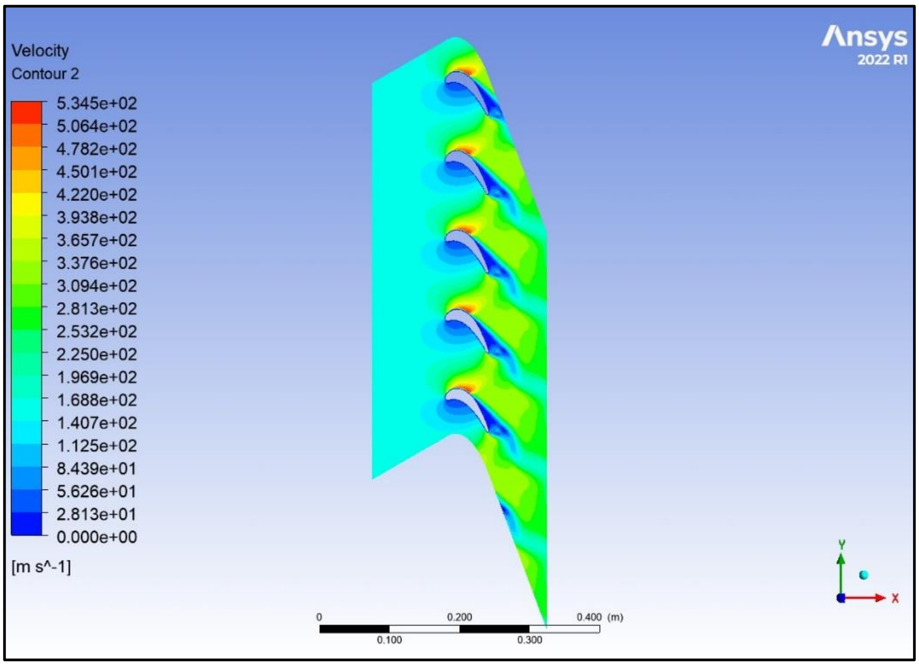


Figure 19; Velocity contour of the blade cascade

From the figure, the separation flow of the first casing is contributing on the upper part of the casing and so on. There is certain possibility that the distance between the two consecutive blade is acting like a convergent nozzle as the cross-sectional area among them is decreasing leading therefore to a flow acceleration. Also, the flow acceleration is due to the highly cambered blades.

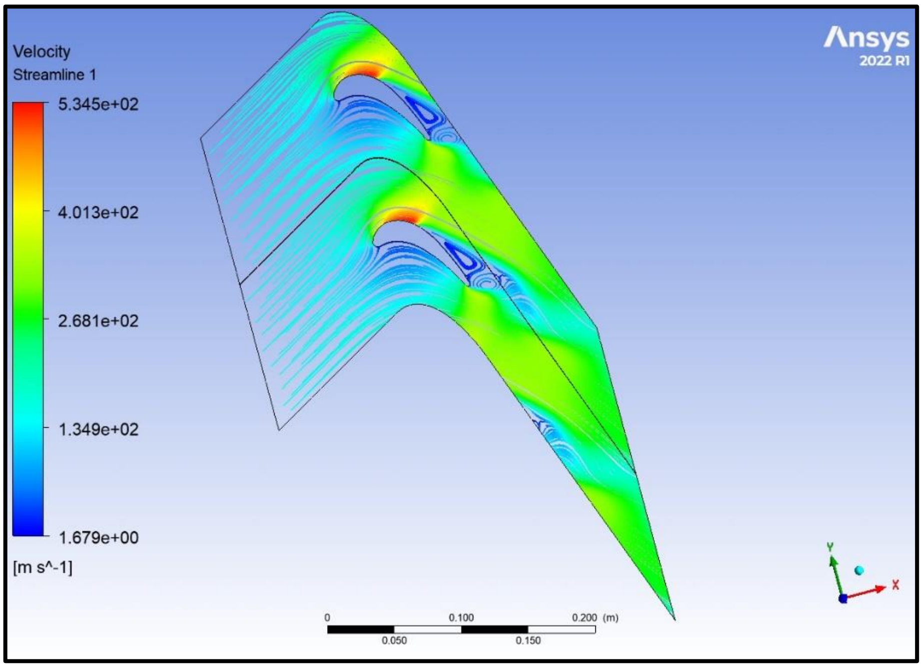


Figure 20:Streamlines over the cascade.

The behaviour of the flow can be better understood by the streamlines. The streamlines help to understand the flow pattern over the blade and casing. It shows how the flow is fitting over the aerodynamic shape of the blade. Furthermore, a vector plot below supports to better visualize the direction orientation and the flow magnitude of the flow. Here, the colours represent the distribution of the velocity, and highlights the vital physical aspects of the flow phenomena.

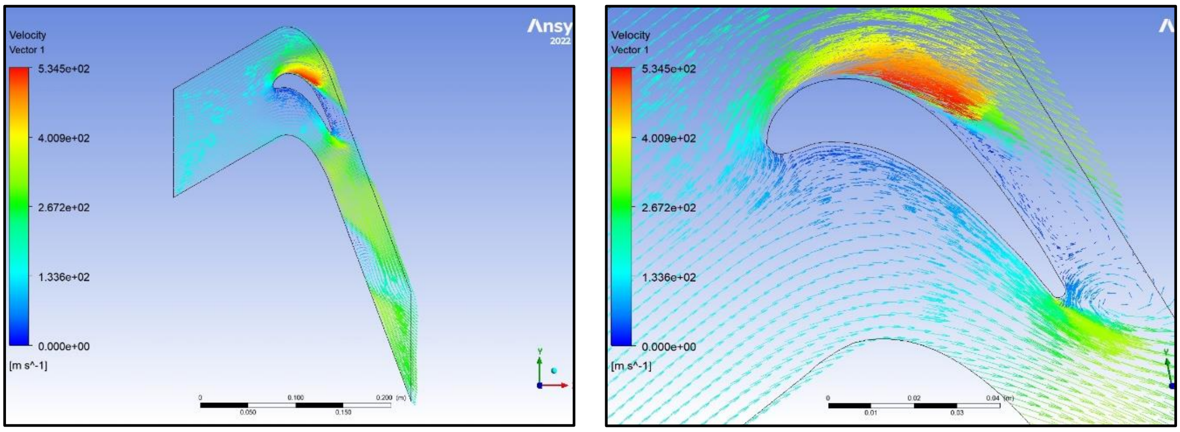


Figure 21: Vector plot of velocity (a) over cascade, (b) over the blade.

As can be observed, the distribution highlights the wake flow region, along with the recirculation vortex as can be visualized by the direction of the vector.

Conclusion:

Overall, the CFD simulation of VKI turbine blade using the CFX solver is performed to determine the performance parameters for a given operating condition. This required a thorough process and an accurate methodology to obtain sound results. Through sensitivity analysis a mesh independency was achieved at around 45258 nodes, thus stable results were attained. Moreover, the isentropic flow distribution over the blade is plotted and total pressure loss is determined. After plotting the contours of pressure, temperature and velocity, the investigation of the flow over the blade showed the creation of wake region on the aft part of the suction surface where the flow begins to separate. Additionally, flow separation highlights the void region with zero velocity and subsequently the recirculation and reverse flow vortex. The separation as caused by an adverse pressure gradient, which resulted into the high-pressure loss affected the performance of the blade. From the aerodynamic point of view, to overcome the problem of the flow separation for the same VKI turbine blade, a change in the angle of attack and optimizing it, is required. With regards to thermodynamics, the total pressure of the flow and total temperature and flow angle at the inlet can be changed.

References:

[1] G. Vatistas, Conservation Equations in Differential Form, Canada, Concordia University: Fluid Mechanics, 2019. W320W9360

[2] Flow in compressors, Von Karman Institute for Fluid dynamics.
Web link: https://www.vki.ac.be/download/research/2012/compressor.pdf

[3] S. Pittala and A.T. Tafere, "CFD Analysis for Linear Blade Cascade of a Turbine", International Journal of Mechanical Engineering, Vol.3, Issue 3, 2014.

[4] V.D. Goryachev, Valery & Ivanov, Nikolay & Smirnov, Evgueni & Ris, Vladimir. "CFD-analysis of Secondary Flows and Pressure Losses in a NASA Transonic Turbine Cascade." ("CFD-analysis of Secondary Flows and Pressure Losses in a ... - SpringerLink") 10.1007/978-3-662-08797-8_21, 2004.

[5] "Ansys CFX-Solver Theory Guide", Ansys Inc., Release 14.0, Nov 2011.
[6] "Introduction to Ansys Meshing", Ansys Inc., Release 15.0, Feb 2021.

Appendix:

Sensitivity Analysis:

The parameters defined in the project are imported from the Ansys workbench:

#	P13 - Mesh Element Size [mm]	P4 - M	P5 - M at Inlet	P6 - M at outlet	P11 - Mesh Nodes	P12 - Mesh Elements	P14 - mass flow [kg s^-1]

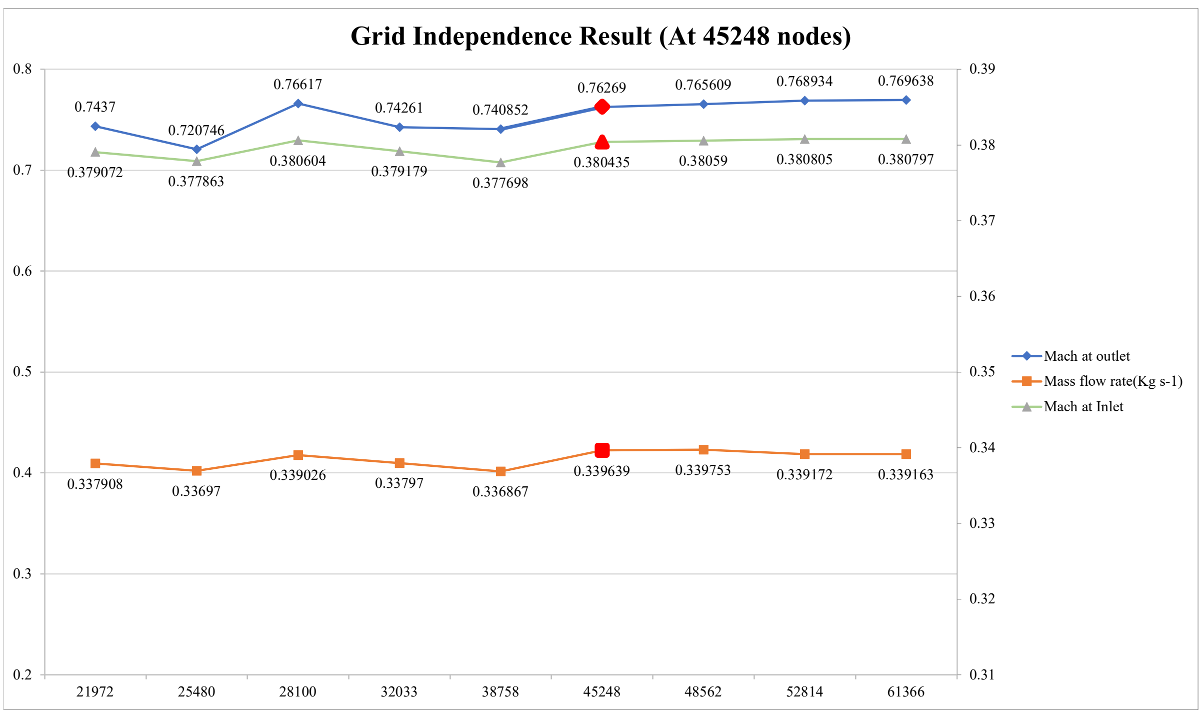
#

The following header line defines the name of the columns by reference to the parameters.

Name	P13	P4	P5	P6	P11	P12	P14
DP 0	8	0.384119	0.379072	0.7437	21972	6355	0.337908
DP 1	7	0.39835	0.377863	0.720746	25480	7339	0.33697
DP 2	6	0.369293	0.380604	0.76617	28100	7596	0.339026
DP 3	5	0.398895	0.379179	0.74261	32033	8538	0.33797
DP 4	4.5	0.395682	0.377698	0.740852	38758	10319	0.336867
DP 5	4	0.410013	0.380435	0.76269	45248	11127	0.339639
DP 8	3.8	0.40354	0.38059	0.765609	48562	11922	0.339753
DP 6	3.5	0.414517	0.380805	0.768934	52814	12692	0.339172
DP 7	3.2	0.425403	0.380797	0.769638	61366	14848	0.339163
DP 8	3.8	0.40354	0.38159	0.75269	48562	11922	0.339753
DP 9	3.8	0.40354	0.38159	0.75269	48562	11922	0.339753

Final Table:

Sno	Mesh Nodes	Mach at Inlet	Mach at outlet	Mass flow out [kg s^-1]
1	21972	0.379072	0.7437	0.337908
2	25480	0.377863	0.720746	0.33697
3	28100	0.380604	0.76617	0.339026
4	32033	0.379179	0.74261	0.33797
5	38758	0.377698	0.740852	0.336867
6	45248	0.381435	0.76269	0.339639
7	48562	0.38159	0.765609	0.339753
8	52814	0.380805	0.768934	0.339172
9	61366	0.380797	0.769638	0.339163



Isentropic Mach Number Calculations:

Chord(C)	Plane distance on blade (x(mm))
80	18.5
80	25
80	30
80	35
80	40
80	45
80	50
80	55
80	60
80	65
80	70
80	75
80	80

Suction Surface:

Pressure [Pa]	Total Pressure[Pa]	Isentropic Mach	x/c
2.38E+05	2.86E+05	0.495165525	0.23125
1.67E+05	2.84E+05	0.858161916	0.3125
1.66E+05	2.84E+05	0.862526851	0.375
1.32E+05	2.80E+05	1.03686286	0.4375
1.35E+05	2.77E+05	1.011939524	0.5
1.20E+05	2.75E+05	1.094266677	0.5625
9.10E+04	2.70E+05	1.27330633	0.625
9.29E+04	2.60E+05	1.235876445	0.6875
1.11E+05	2.31E+05	1.024463341	0.75
1.18E+05	2.01E+05	0.86238581	0.8125
1.20E+05	1.90E+05	0.79627875	0.875
1.25E+05	1.99E+05	0.801106095	0.9375
1.38E+05	1.85E+05	0.622755624	1

Presure Surface

Pressure [	Total Pressure[Pa]	Isentropic Mach	x/c
2.38E+05	2.86E+05	0.495165525	0.23125
2.78E+05	2.88E+05	0.209979614	0.3125
2.79E+05	2.88E+05	0.203174612	0.375
2.77E+05	2.88E+05	0.216627067	0.4375
2.77E+05	2.88E+05	0.215576553	0.5
2.77E+05	2.88E+05	0.222037991	0.5625
2.72E+05	2.88E+05	0.267807495	0.625
2.68E+05	2.87E+05	0.299581304	0.6875
2.58E+05	2.87E+05	0.374178822	0.75
2.45E+05	2.87E+05	0.457314923	0.8125
2.25E+05	2.88E+05	0.575639509	0.875
1.77E+05	2.52E+05	0.691564518	0.9375
1.38E+05	1.85E+05	0.622755624	1