Volunteering Internship Project Report

Numerical Simulations of the Turbine Cascade

at

Institute of Thermal Turbomachinery

Karlsruhe Institute of Technology, Germany

Submitted by

Lovish Chopra

Under the guidance of

Jonas Schmid

1st May 2021 - 31st July 2021

1. Methodology

First, the 2-D Geometry at mid-way location is extracted from 3-D geometry in CATIA V5 of the turbine blades provided by ITS – KIT in as shown in the figure below: -

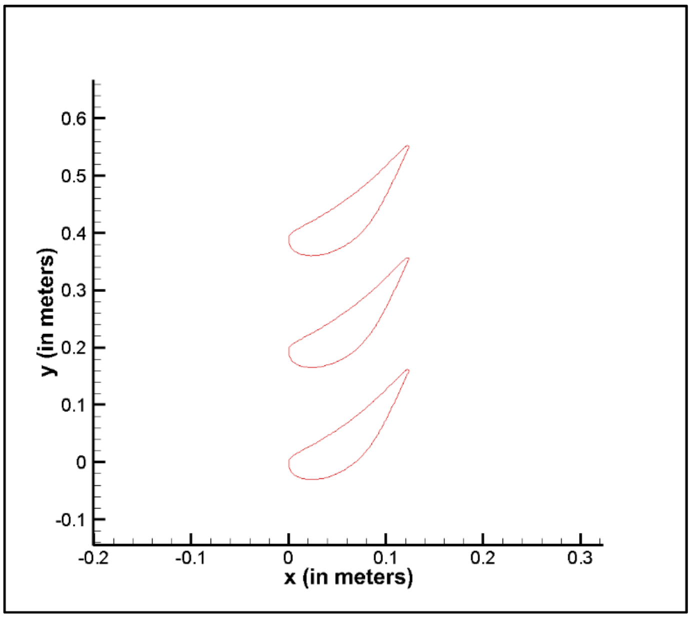

Figure 1: Blade geometrical configuration

Further domain is created in Pointwise with 1.5 chord times upstream and 2.5 chord downstream. A total of four passages are considered which consists of three main vanes with a upper vane and lower vane as shown as follows,

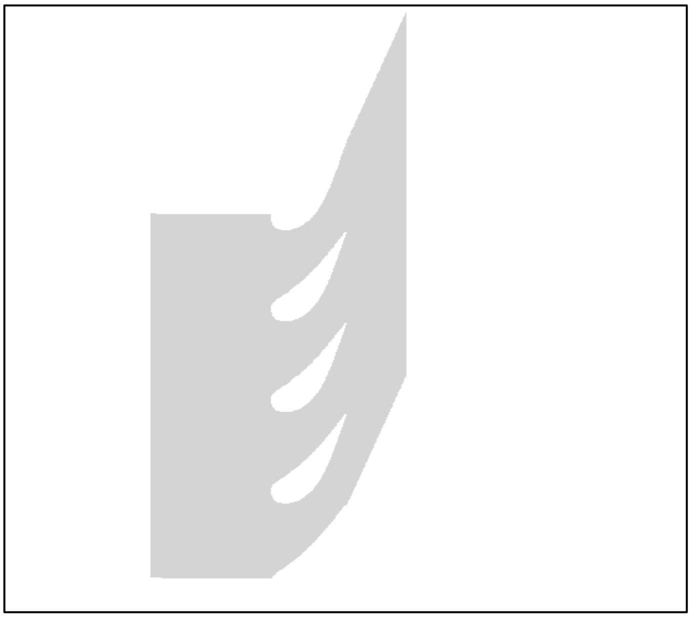

Figure 2:Domain Illustration

For the domain discretization, a grid independence study concluded a grid size of 2 mm (114811) to be a nominal one ran with a test case using the solution procedure as described in further description in comparison to various grid sizes of 44105 (3mm),60000(2.5),114811 (2mm),172000 (1.5mm) and 297000(1mm). Mesh generated is presented in Figure 3 followed in Grid sensitivity Analysis study in Figure 4. It should be noted that value of Y^+ factor is kept less than 1 in order to capture proper wall effects throughout the computational domain.

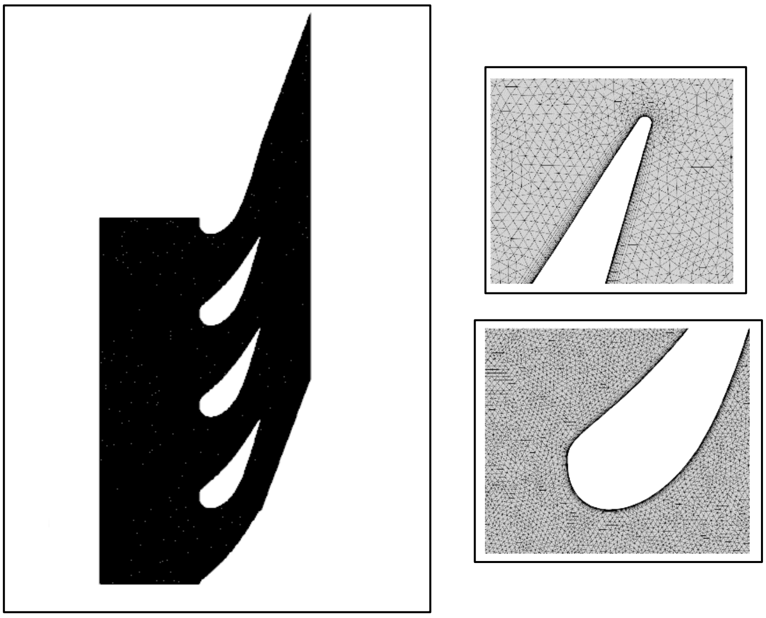

Figure 3: Domain Discretization

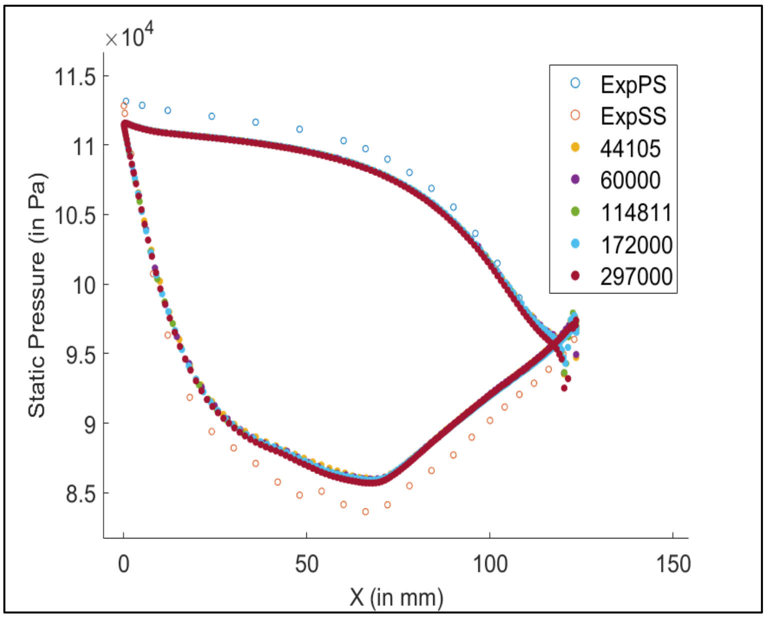

Figure 4:Grid Independence Study

It has t has to be noted from this point out of simulation data of 2nd main vane will be used to compare with the experimental results. Density Based Navier – Stokes solver is incorporated in Ansys Fluent with the help of Shear Stress Transport $k - \omega$ as the turbulence model. Type of boundary conditions used in this work are illustrated in figure below, Pressure inlet with a temperature of 425 K is used with a backflow of 300 K. 2nd Order accurate first order central differencing scheme is used for momentum and pressure calculations. Implicit formulation was used in this work with initialization method as hybrid for both vectors and scalars. Convergence Criteria was set to 0.001 for momentum continuity, and energy equations.

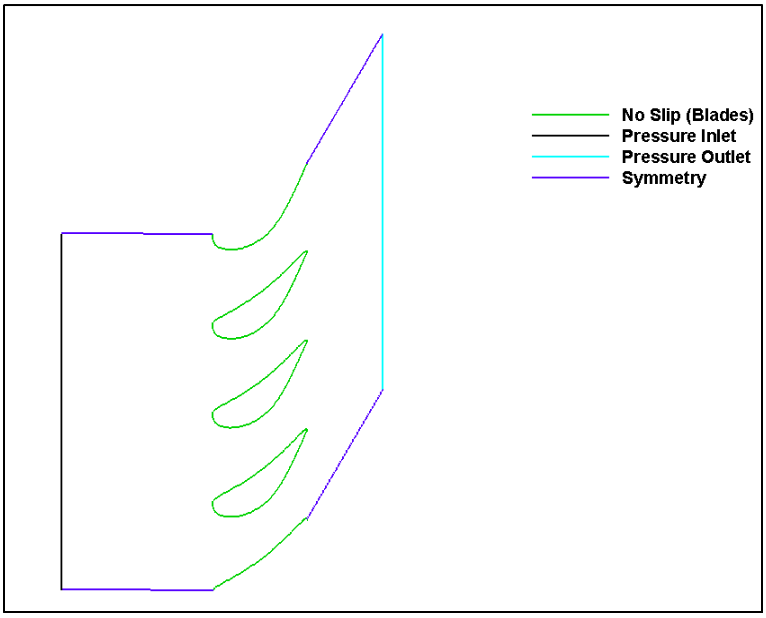

Figure 5: Boundary Conditions types Specified

2. Validation

For the validation purposes so as to match best with the experimental results, values of Inlet Pressure and outlet pressure are obtained through an iterative process. In this study, first sensitivity analysis for inlet pressure is carried out for a particular outlet pressure as shown in the Figure 6 and 113100 is selected for further sensitivity analysis for selecting outlet pressure. Consequently, a similar study is carried for outlet pressure in aim to get a best value. It is illustrated in Figure 7.

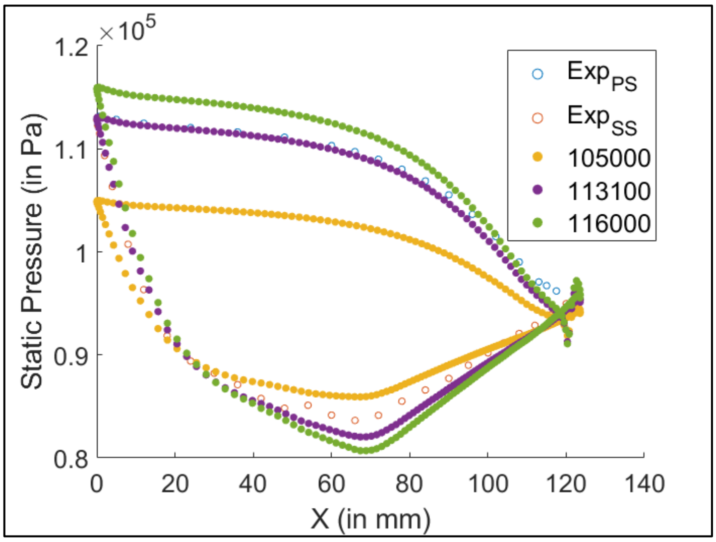

Figure 6: Sensitivity Analysis for Inlet Pressure

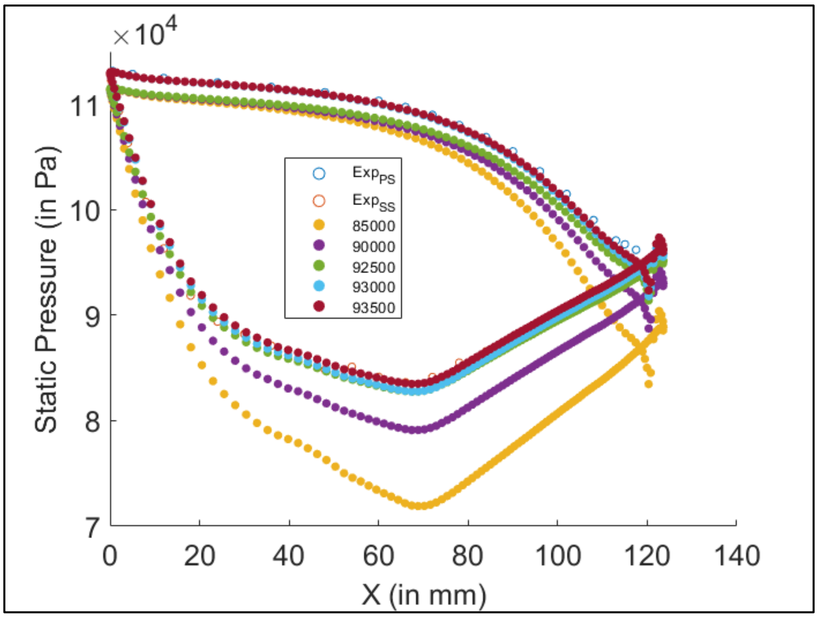

Figure 7: Sensitivity Analysis for Pressure Outlet

3. Results and Discussions

As concluded from the validation, set of values with Inlet Pressure of 113100 Pa and an outlet pressure of 93500 Pa is finalized. Results are in excellent agreement with the experimental results. However, a slight underprediction can be observed just at the pressure side near the trailing edge as represented in Figure 8.

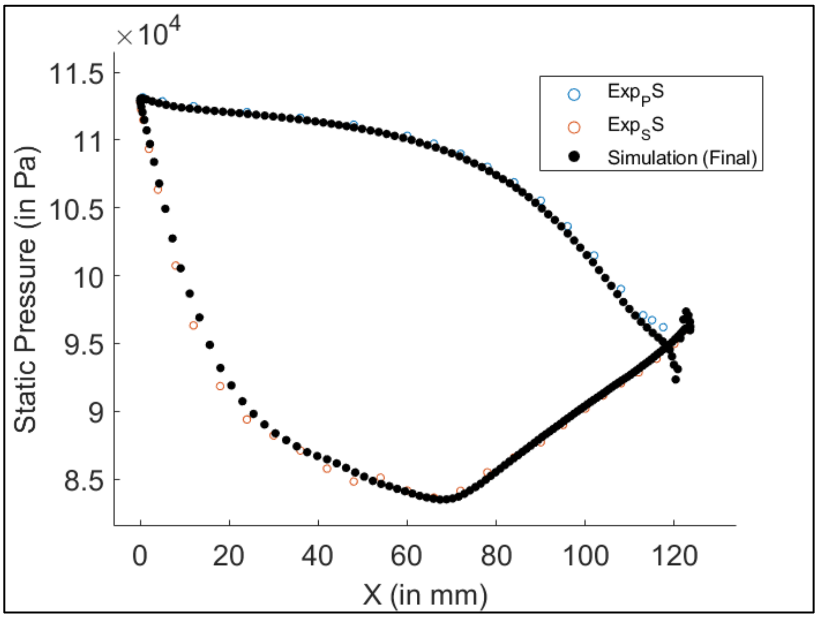

Figure 8: Final pressure distribution

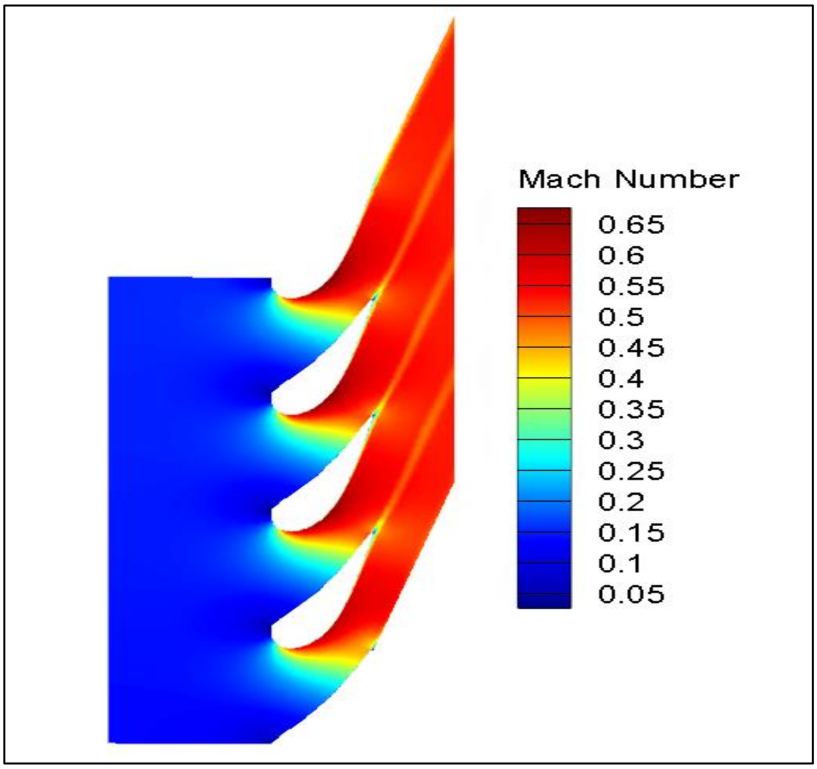

Figure 9: Mach Number distribution through the passages

Figure 9 represents Mach number distribution across all the passages. Slight prediction is noted with 4-6 % Error. Same can also be observed in plots for temperature and pressure as well.

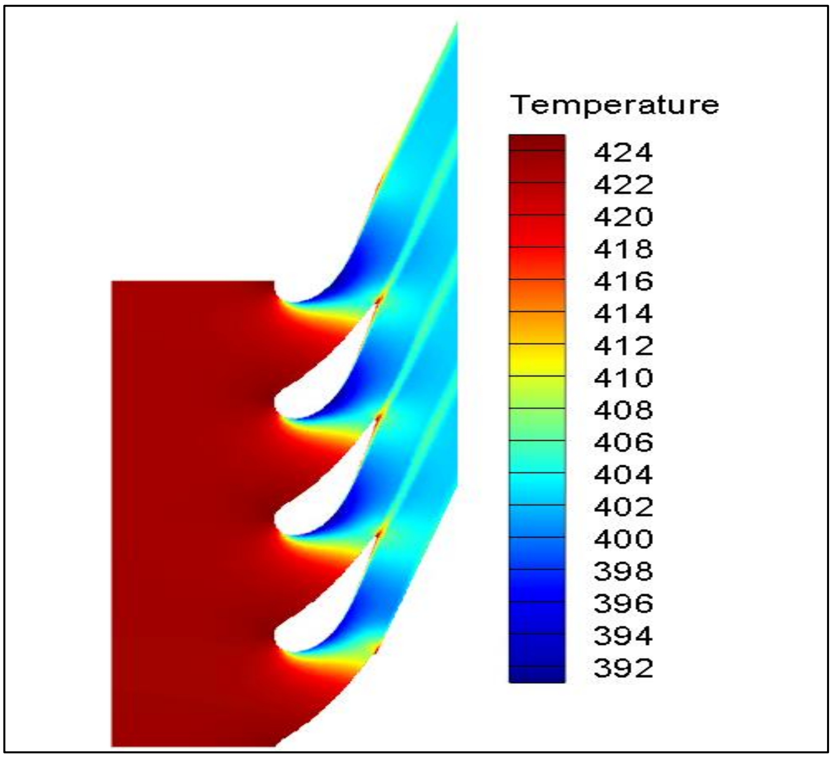

Figure 10:Temperature distribution through all passages

Streamlines are also presented so as to gain insights of the flow direction.

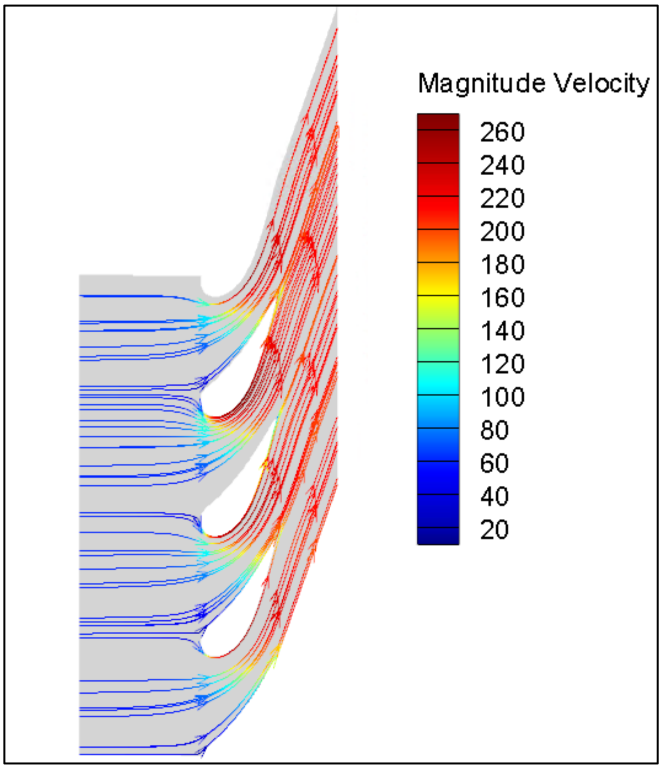

Figure 11: Velocity Streamlines overview

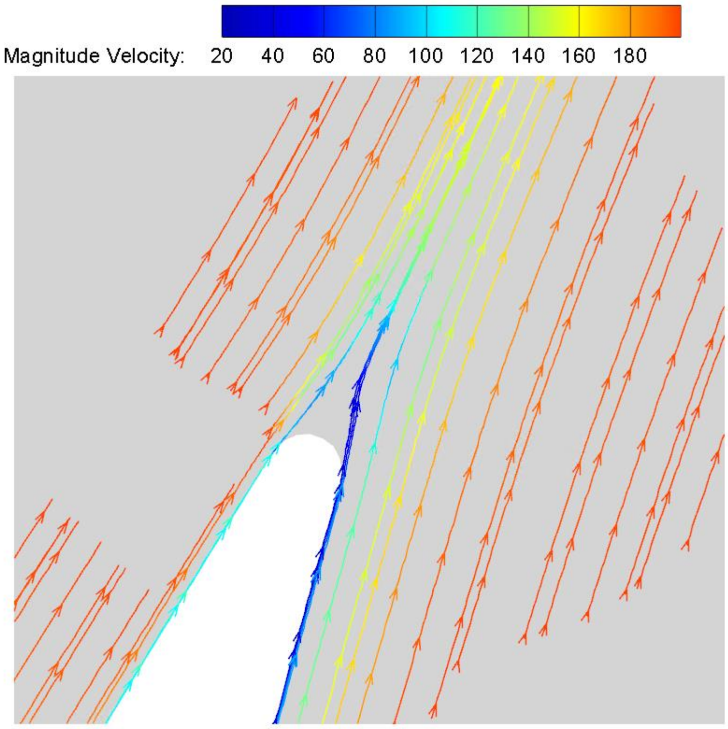

Figure 12: Streamlines near the trailing edge

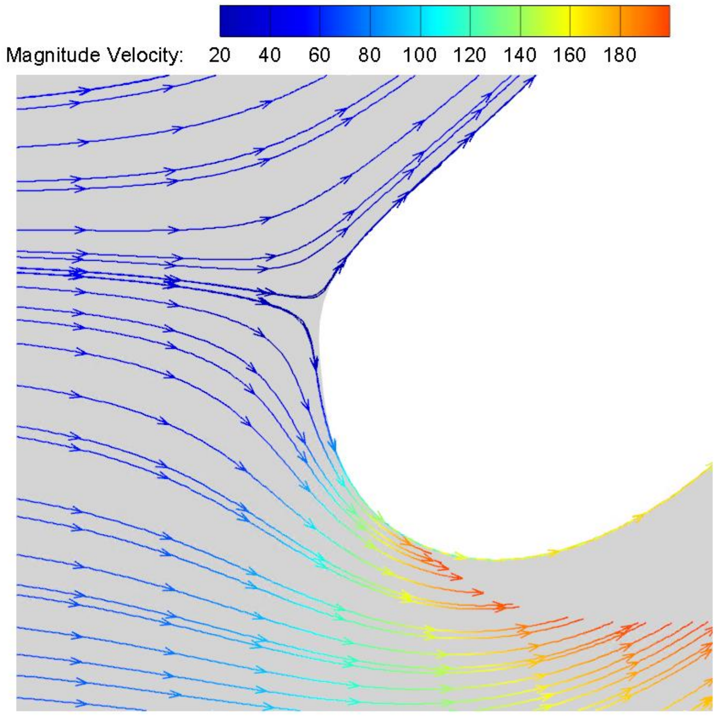

Figure 13: Streamlines at leading edge

4. Conclusions

In this work, 2D CFD simulations for the turbine passage at ITS-KIT have been carried out. Results for the main vanes are in good agreement with the experimental ones.

Future implementation with this work shall be carried out by comparing these 2D simulation results with the 3D simulated results. Moreover, for enhanced