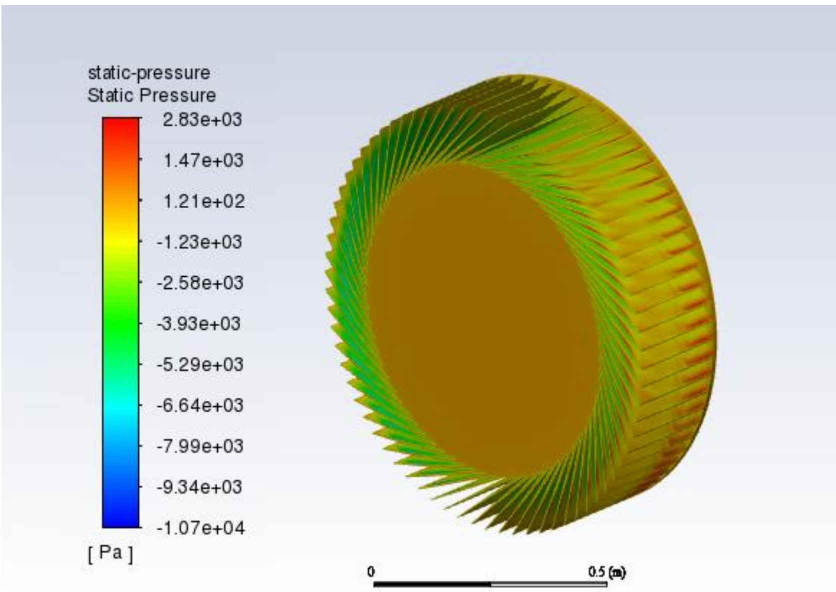

CFD ANALYSIS FOR PF900 WITH 72 TAPERED FINS AT 15 DEGREE AND 232MM THICK

Working Principle and PF Efficiency

ABSTRACT

A circular foil connectable to a source of power for producing thrust in axial direction when rotated by generating vacuum at one side while maintaining the atmospheric pressure on the other side.

Dr. M. Aqeel Rashad
Ph.D *(CFD Simulation of Jet and Vortex Actuator with and without Crossflow)*
University of Stuttgart, Germany

Nabil Arslan
B.Sc Mechanical Engineering
University of Engineering and Technology, Lahore, Pakistan

Subject:

CFD Analysis for PF900 with 72 Tapered Fins at 15 Degree and 232mm thick.

Scope:

The scope of this study is to analyze the working principle and find the Efficiency of the Rotating Thrust-Producing Apparatus (Pressure Foil) which is a US Patent against patent number 5,328,333.

Background:

We were assigned a task on Freelancer.com by Mr. Steve Quinn (owner of the patent) to analyze the apparatus described above for its working principle and compare the PF efficiency obtained with the results of analysis of same apparatus on another platform 'Simscale'.

Geometry and Mesh:

Geometry and mesh of the apparatus is shown in the following figures:

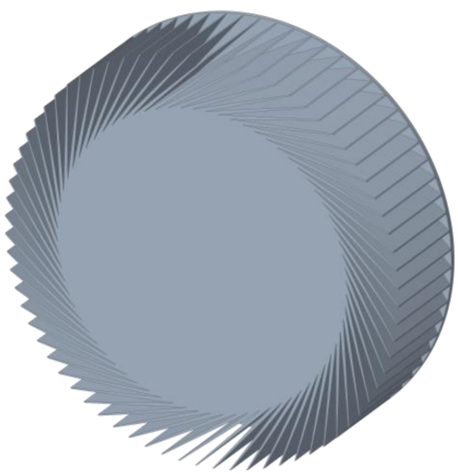

Fig-1a: Geometry

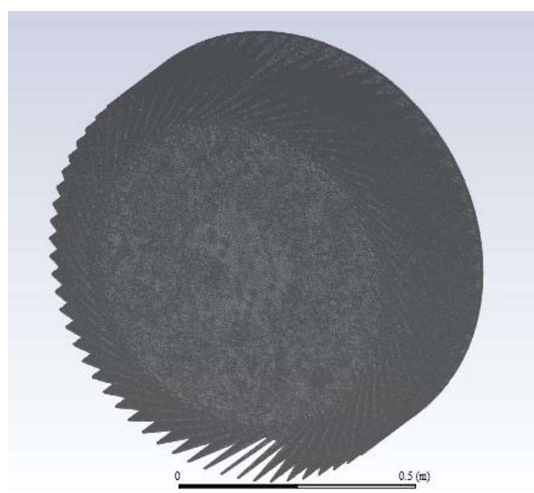

Fig-1b: Mesh

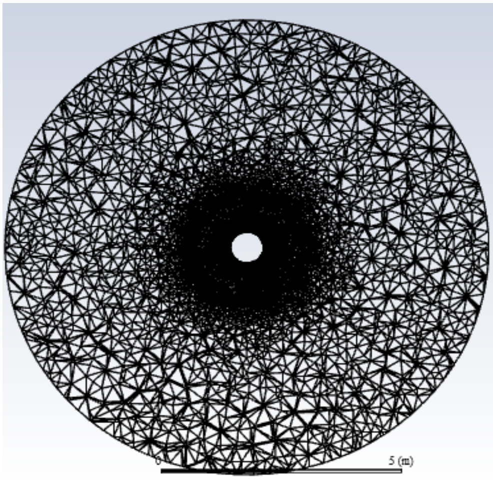

Fig-1c: Meshed Domain

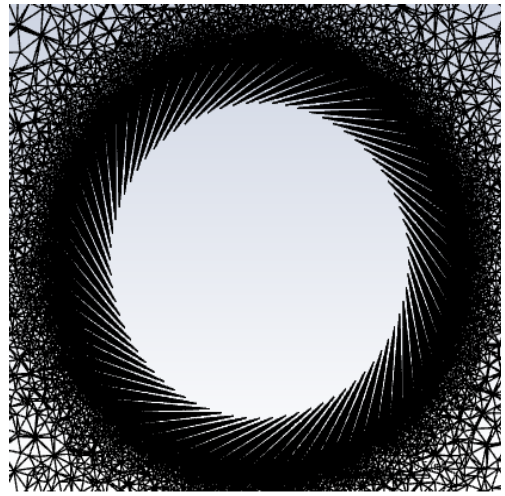

Fig-1d: Center of Meshed Domain

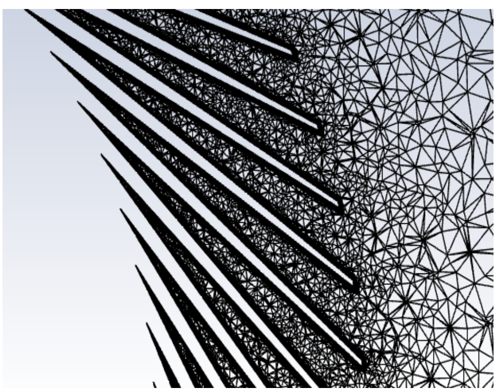

Fig-1e: Close up of fins

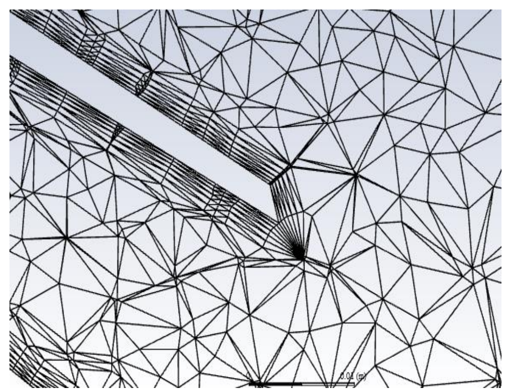

Fig-1f: Boundary Layer at fins

In order to carry out the analysis, a 10m long cylindrical fluid (air with default properties) domain was created having a diameter of 5m. The rotating apparatus is at the center of this domain. The domain consists of two parts ‘Inner’ and “Outer”. Inner domain is rotating at a specified speed and outer domain is stationary. Total numbers of cells generated were about 9.8 million including six boundary layers to capture the viscous effect.

Boundary Conditions:

Velocity inlet with ‘0’ magnitude and ‘0’ initial gauge pressure was applied at inlet while pressure outlet with ‘0’ magnitude was applied at outlet of the domain.

Turbulent specification method for both boundary conditions was based on K and omega with turbulent kinetic energy of 0.00375 m^2/s^2 and specific dissipation rate of 3.375 s^{-1}.

Moving wall boundary condition was applied at the apparatus’ walls.

Flow/Solution Setup:

A 3D, incompressible steady state simulation with moving reference frame (MRF) using SST K-omega turbulent model was run.

SST K-omega is a hybrid two equation model to capture the near wall effects and behavior away from the wall which is best suited for our simulation.

Air is used as fluid medium with default properties. Pressure velocity coupling with coupled scheme and for discretization, second order momentum, pressure, turbulent kinetic energy and specific dissipation rate was used.

Results and Discussions:

In this section simulation results and findings are presented and discussed.

The device used in this study is shown in Fig-1a and explained in the US patent number 5,328,333.

This device (PF) is used for producing axial thrust by rotation it in a fluid such as air or water. The working principle of this device is based on pressure difference created between front and rear face of the device by rotating fins. This pressure difference between the front and rear face generates a forward thrust.

In this study this device is rotated at angular speed to estimate thrust generated and cause of this thrust (working principle).

Pressure contours on this device are shown in Fig-2a to Fig-2c. These contours show that at the rear plate there is atmospheric pressure (in this case zero Pascal). At the front plate, again pressure is atmospheric however, at the fins top surface and between the fins, negative pressure (vacuum) is created which in turn created a pressure imbalance. This pressure imbalance results in forward thrust.

What we noticed in this study is that when we rotate this device at sufficiently high speed it releases fluid (air) from spaces between the fins at a faster rate than they can be replaced. Fluid release from the fins go downstream at high speed and in reaction fluid at the back of this device starts to flow towards back plate. This can be seen in velocity contours in Fig-3a and 3b. We can see in these figures that the velocity of fluid at back plate is very less than that of the fluid flowing downstream the fins. So, we can say that fluid at the back plate is sticking to it and hence producing a positive pressure while the fluid in the gaps between the fins is moving very fast and hence creating a negative pressure. This creates an imbalance of fluid pressure on the back and front side of this device as a result this device gives thrust in the direction (forward) of low pressure.

In numerical/quantitative terms, in this study, the device is rotated at angular speed of 2000 rpm (~210rps) for 5700 seconds. The value of axial thrust (pressure force in z direction) and resisting/counter moment (about z-axis) began to stabilized after 1200 seconds around '311 N' and '-231 N-m' respectively.

Formula for the PF efficiency has units newton per kilowatt (N / KW) which has the form as below:

PF efficiency = Thrust (N)/ [Moment (N-m) x Agular velocity (rps)]

As per this formula, the PF efficiency calculated is 6.4 N/KW.

PF efficiency = 311 (N) / {(231(N-m) x 210 (rad/s)} = 0.0064 N/Watt or 6.4 N/KW

Conclusion:

As evident by this study that rotating this device at a sufficiently high speed creates an imbalance of pressure between front and rear side and as a result of this pressure imbalance a forward thrust is produced.

Fins at the front are the main source of pressure imbalance therefore, if we optimize the shape of fins and area in such a way that it produces more vacuum and minimum rotating drag, the efficiency of this device can be increased further.

The results of this study compliment the claims made in the reference patent, the 'Simscale' study and that of experimental results communicated by the owner of the patent.

Future Work:

This device will be optimized to increase its efficiency by keeping overall diameter constant and changing the fin's size and shape.

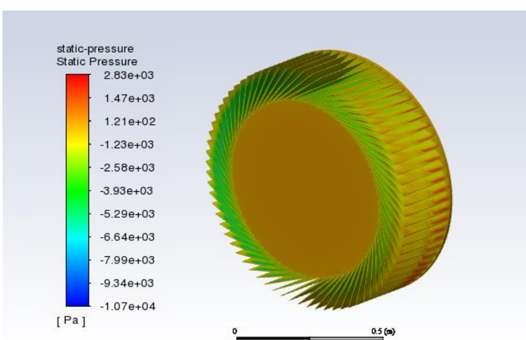

Fig-2a: Static pressure at PF front face

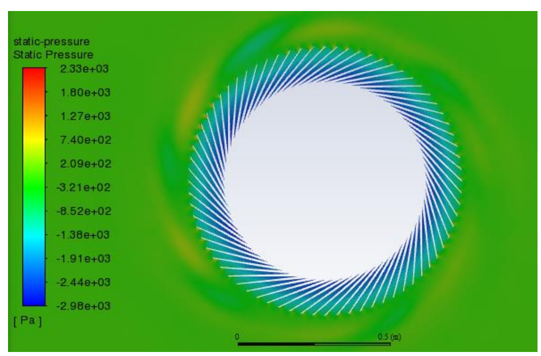

Fig-2b: Static pressure at surface b/w fins

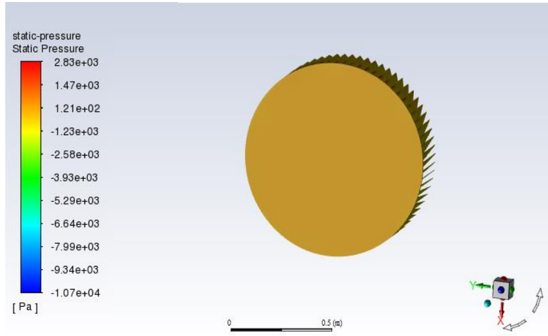

Fig-2c: Static pressure at rear face

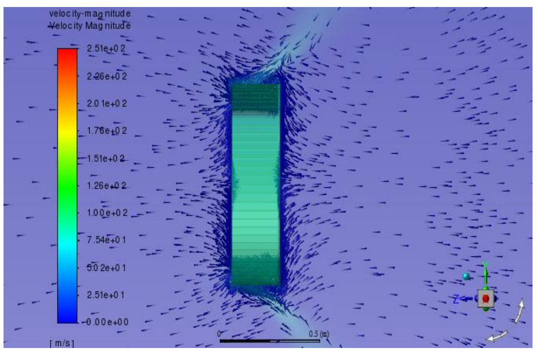

Fig-3a: Velocity magnitude

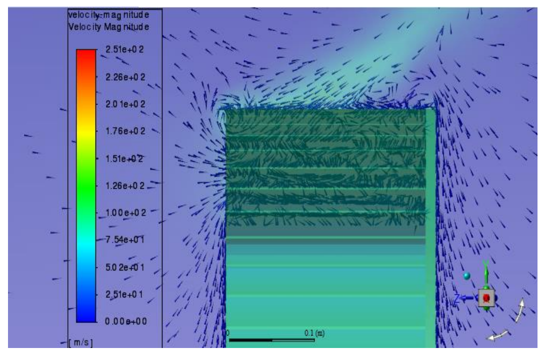

Fig-3a: Velocity magnitude (zoom)