Journal of Mechanical Engineering Research and Developments (JMERD)
DOI : http://doi.org/10.26480/jmerd.03.2018.91.96
ISSN: 1024-1752
CODEN : JERDFO

IMPROVING AND ANALYSIS TURBINE WHEEL OF TURBOCHARGER FOR HIGH-PERFORMANCE ENGINES

Mohand Mosa Mohammad[1], Ahmed Esmael Mohan[2], Hiyam Adil Habeeb[2]

[1]***Mussaib* Technical Institute, Al-Furat Al-Awsat Technical University, Babylon, Iraq.**
[2]***Technical Collage/ Al-mussaib,* Al-Furat Al-Awsat *Technical University, Babylon, Iraq.***
Corresponding Author Email:

ARTICLE DETAILS

ABSTRACT

A turbocharger performance of exhaust gas turbine on the diesel engines are studied the turbine driven compressor that is used for increasing air flow rate into internal combustion engine the angle of Blade effect on the flow field of turbine wheel. Angle of Blade of turbine wheel analyzed Based on the theory Computational Fluid Dynamics (CFD). How the various blade angle of turbine wheel due to optimization on flow field along the blade and leads to optimization in the pressure and velocity. Blade angle studied is conducted for understand the flow behavior in order to minimize the effects blade angle. in this paper dealing The Analyses the flow field along the blade passage of the turbine wheel for the turbocharger and study the effect flow field at different blade angle for turbine wheel outlet blade angle, β = 65⁰, 45⁰ and 35⁰.

KEYWORDS

CFD, Turbo charger, turbine wheel, Blade angle, Pressure contour, Velocity streamline

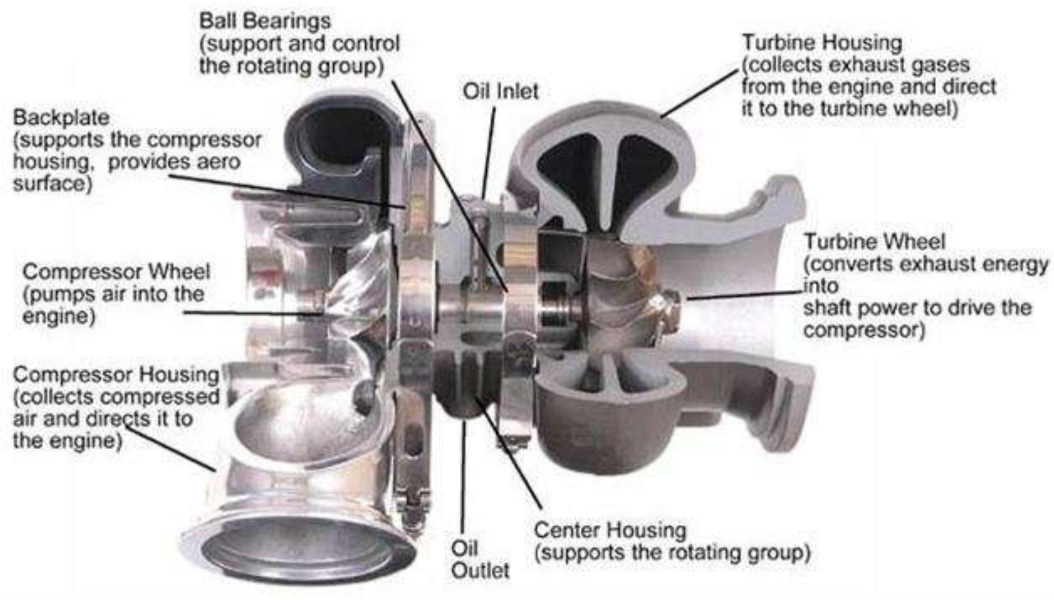


Figure 1: Compressor and turbine sections of turbocharger

1. INTRODUCTION

The turbine driven compressor that using foe increase the air flow rate into internal combustion engine. This is accomplished by used the compressor for compress the air and increasing the air flow into the cylinders of the internal combustion engine. The compressor is driven by a turbine that extracts energy from the exhaust gas for the internal combustion engine and after that converts that energy to shaft power. The arrangements are ideal because a compressor is not using power from the engine drive shaft. Turbocharger is efficient in that the turbine utilizes the exhaust gas energy that is typically reject to the atmosphere on the case of a naturally aspirated engine. For turbocharger for operate efficiently.

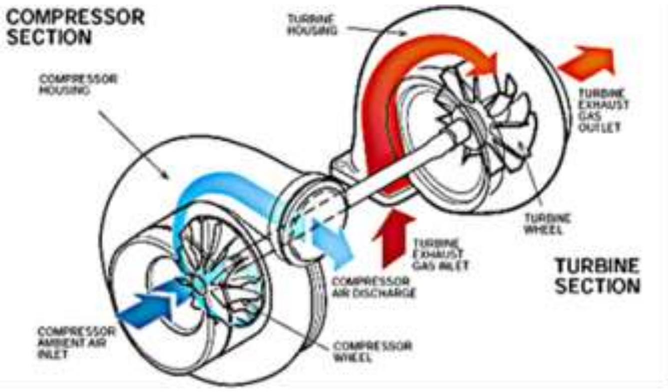


Figure 2: Turbocharger

An important problem that arises in the design and the performance of the wheels of a turbocharger is the understanding, analysis and prediction of internal flow on the wheels of a turbocharger. The two available means of evaluating new wheels of a turbocharger design is computational (cfd). The advantages of computation simulation (CFD) are low cost, increase speed, and more data,

Through the use of advanced and accurate computer simulations, much of the preliminary experimental testing may be eliminated.

2. SOLUTION METHODOLOGY

Design and Analysis of a Turbine Wheel Turbocharger: using PRO-E and ANSYS_CFX. In ANSYS _CFX the design of the turbine wheel is done using PRO-E with following:

1.2 Design and Analysis of a Turbocharger Turbine Wheel

Geometrical Specifications:

Inducer diameter D_1	= 141.2mm
Exducer diameter D_2	= 124mm
Trim	= 77.12
Tip height	= 20mm
Blade height, H	= 61.06mm
No. of blades of Turbine, N_b	= 12
Thickness of the blade, t	= 2 mm

The above-mentioned geometrical specification is same for 65^0,45^0 and 35^0 outlet blade angle of a Turbine wheel. The analysis is done for each of above specified outlet blade angle and is as follows:

Modeling**:** Designing it done by PRO- E. turbine whell with above geometrical specifications with 2mm thickness throughout the blade (Figure 2).

Case – I Outlet Blade Angle β = 65^0

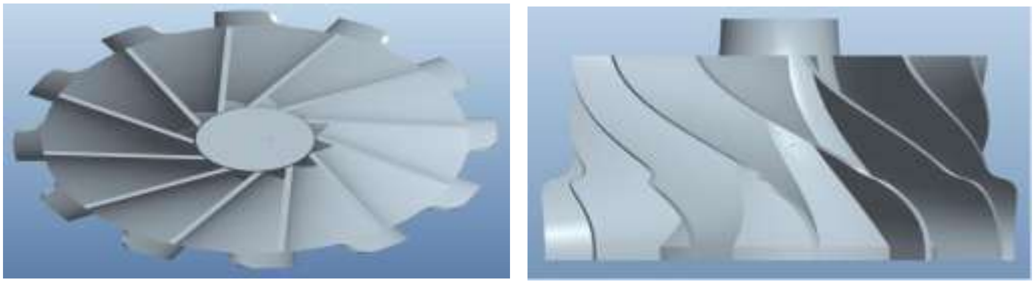


Figure 3: Turbine Impeller (wheel) at β = 65^0

2.1.1 Mesh Generation

ANSYS mesh generation is powerful tool that lets designer of rotating machinery to create high quality Tetrahedral meshes, these mesh is using in the (ANSYS) for solve the complex blade problem.

Fine mesh information and tetrahedral Meshing done is shown in Table 1 and mesh generation along with section view figure below.

Table 1: Mesh Information for Outlet Blade Angle B = 65^0

Fluid	81814	410994
Impellor	24092	108184
All Domains	105906	519178

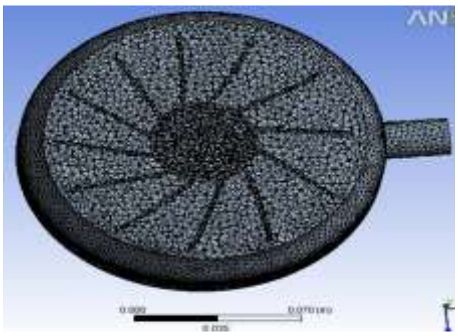


Figure 4: Tetrahedral mesh generation

2.2 The Boundary Conditions

After creating mesh in ANSYS, the mesh is analyzed in ANSYS CFX-Pre for applying boundary condition. And boundary condition is same for outlet blade angle 65^0 ,45^0 and 35^0 of a turbine wheel of turbocharger. Boundary conditions the flow and thermal variable on the boundary of the applied boundary conditions shown in Table 1 and Table 2 and is same for 65^0 ,45^0 and 35^0 outlet blade angle of a turbine wheel. Boundary condition is considered in CFX. The working fluid is taken as CO_2 at 1200 ^{0}C and reference pressure at 1 atm. The fluid domain is considered as stationary. ANSYS_CFX software is used for obtaining the results and standard k-ε turbulence model with turbulence intensity of 5% considered

- **Inlet boundary condition:** impeler inlet boundary condition is set in a sub-sonic flow regim with total pressure at 1atm. The turbulence intensity is 5% considered.

- **Outlet boundary condition:** The impeler outlet boundary condition is set in the subsonic flow regim with relative pressure 0 kpa.

- **Solid boundary:** The solid or wall, boundaries include the Turbine. A smooth surface and no heat transfer (adiabatic flow) were assumed for all the wall boundaries**.**

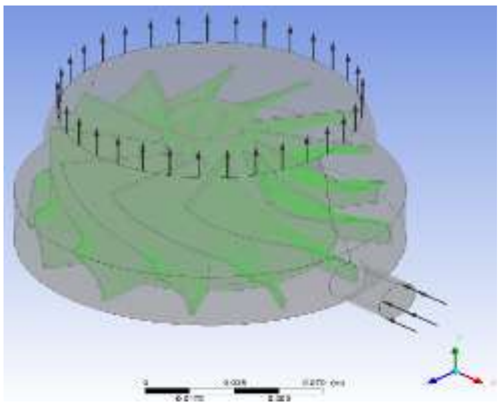


Figure 5: Boundary condition

Table 2: Domain Physics For outlet Blade Angle β = 65^0

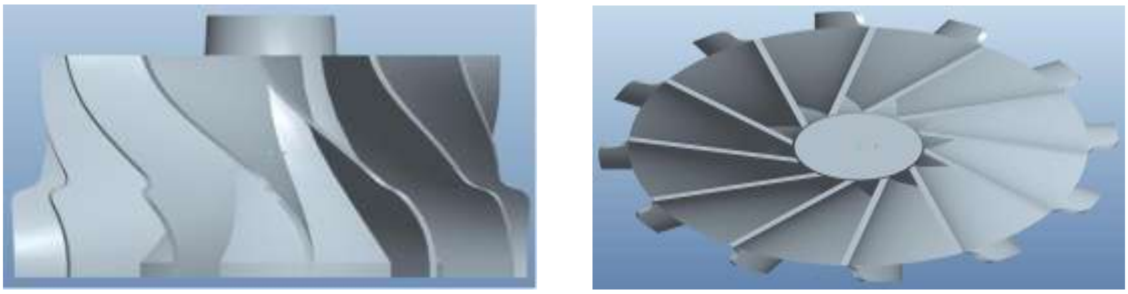
Domain - Fluid	
Reference Pressure	1.0000e+00 (atm)
Stiffness Model Exponent	1.0000e+01
Heat Transfer	Isothermal
Temperature of Fluid	2.5000e+01 (C)
Turbulence Model	[k epsilon]
Domain - Impellor	
Angular Velocity	1.5000e+04 [rev min^-1]

Table 3: Boundary Physics for Inlet Blade Angle β = 65⁰

Domain	Boundaries	
Fluid	**Boundary - Inlet**	
	Settings	
	Flow Regime	Subsonic
	Normal Speed	1.5000e+02 [km hr^-1]
	Turbulence	Medum Intensety and Eddy Viscosity Ratio

2.2.1 Case – II outlet Blade Angle, β = 45⁰

Modeling: Designing is done with the help of a PRO- E. turbine impeller with above geometrical with 2mm thickness throughout blade.



(a) Front view (b) Top view

Figure 6: Turbine wheel at $\beta_2 = 45^0$

2.2.2 Mesh Generation

ANSYS mesh generation is powerfol tool that lets designer and analyst of rotating machinery to create high quality Tetrahedral meshe, while preserving the underlying geometry. These meshes is using in the (ANSYS) to solve blade problems. Fine tetrahedral Meshing done and mesh information and mesh generation along with section.

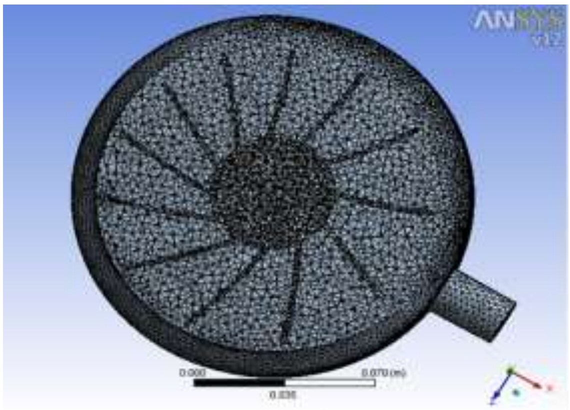


Figure 7: Tetrahedral mesh generation

Fine tetrahedral Meshing done, and mesh information is shown in Table 4 and mesh generation along with section.

Table 4: Mesh Information for Outlet Blade Angle B = 45⁰

Domain	Nodes	Elements
Fluid	60716	326219
Impellor	29231	119355
All Domains	89947	445574

2.3 Boundary Conditions

After Creating a Mesh In Ansys, The Mesh Is Analyzed In The Ansys by Applying the Boundary Condition The Boundary Conditions Is Similar To Outlet Blade Angle, **β** = 65⁰ Of A Turbine Wheel.

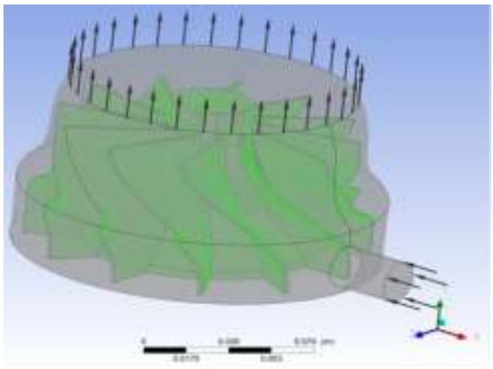


Figure 8: Boundary condition

2.3.1 Case – Iii Outlet Blade Angle, B = 35⁰

Modeling: The Designing is done with help of PRO- E. turbine impeller with above geometrical specifications with 2mm thickness throughout the blade.

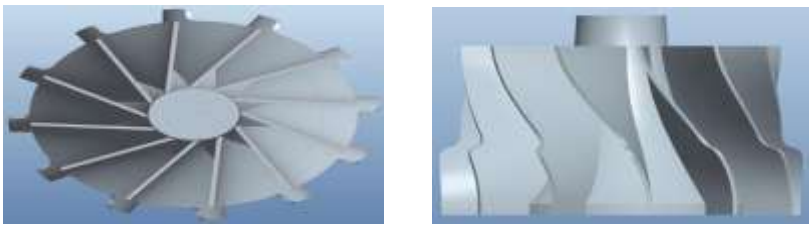


Figure 9: Turbine wheel at β = 35⁰

2.3.2 Mesh Generation

Fine tetrahedral Meshing done, and mesh information is shown in Table 10 and mesh generation along with section view (Figure 10).

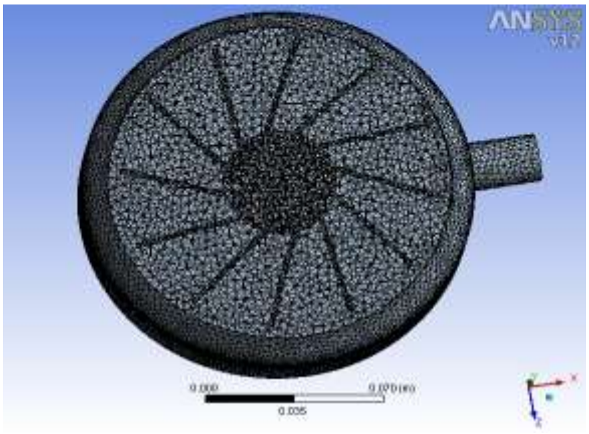


Figure 10: Tetrahedral mesh generation

Table 10: Mesh Information for Outlet Blade Angle $\beta_2 = 35^0$

Domain	Nodes	Elements
Fluid	61028	328078
Impellor	27972	115462
All Domains	89000	443540

2.4 Boundary Conditions

After creating the mesh in ANSYS, the mesh is analyzed in the ANSYS CFX-Pre for applying boundary conditions. The boundary condition is similar to outlet blade angle, $\beta = 65^0$ of a turbine wheel.

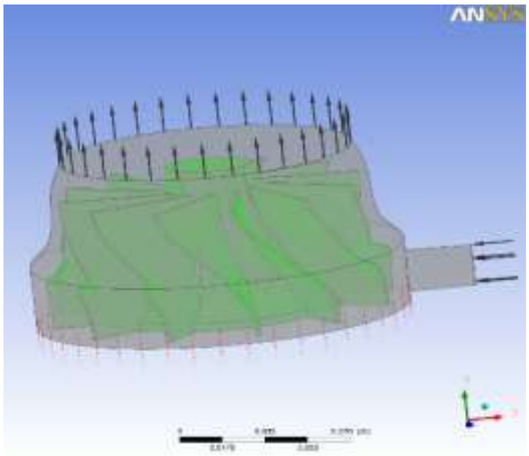


Figure 11: Boundary condition

2.5 Solver Control

CFX Solver Manager is visual interface that display the variety of the result and must be used when plotted data needs to be view during problem solving. The turbulence was simulated with k- ε turbulence model. The advection scheme was set to upwind. The convergence criterion was set to the residuals smaller than 10^{-4}. The fluid time scale is set to as physical time scale which is 0.00002s. By using these values and changing the blade angle, the model is solved and various contours are taken.

3. RESULTS AND DISCUSSION

Analysis of a turbine wheel for different blade angles of the turbocharger is done, to study the behavior of velocity streamline and pressure at speed 150km/hr and wheels of a turbocharger is at 15000 (rpm).

3.1 Analysis of The Turbocharger Turbine Wheel

The Analysis of Turbine wheel at different outlet blade angles such as 65^0, 45^0 and 35^0 is done and discussion for each outlet blade angle as follows:

3.1.1 Case – I Outlet Blade Angle, $B_2 = 65^0$

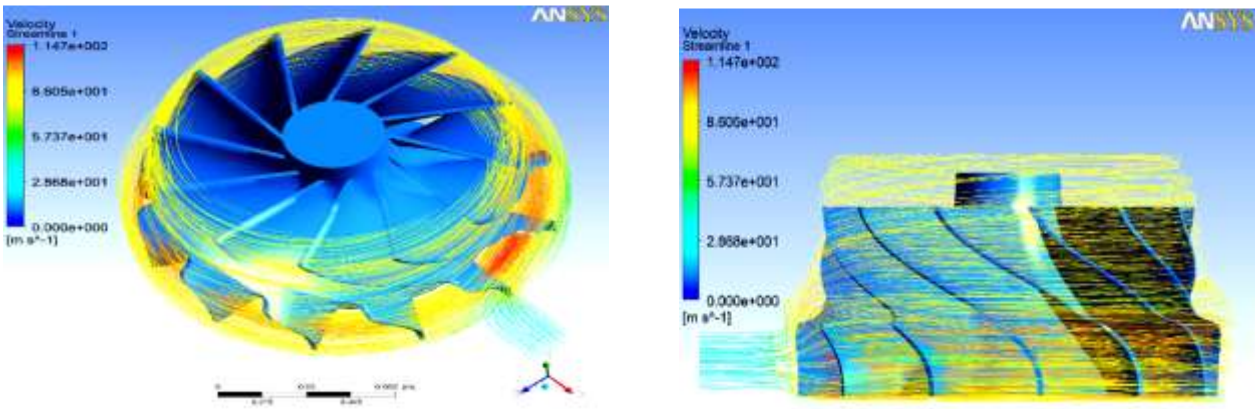
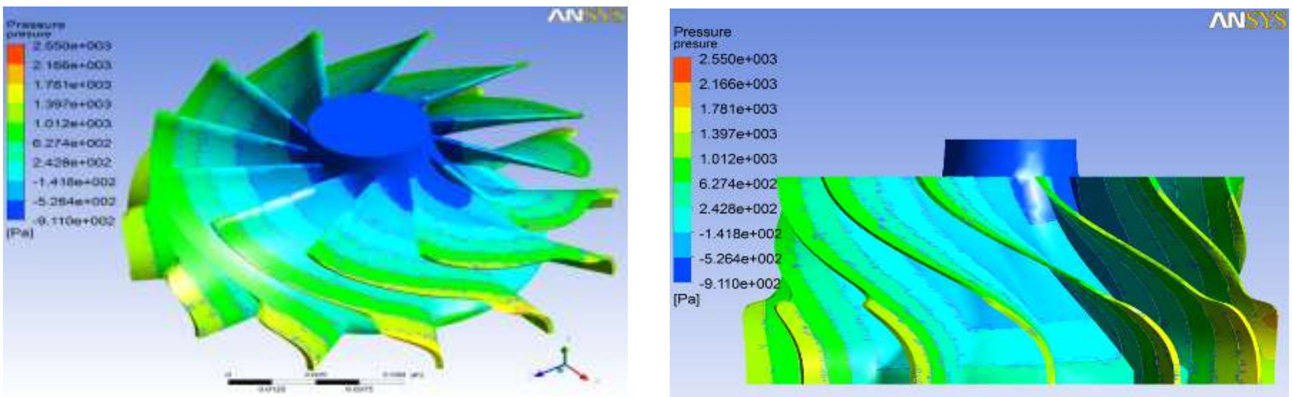


Figure 12: Velocity streamline at $\beta = 65^0$



Cite The Article: Mohand Mosa Mohammad, Ahmed Esmael Mohan, Hiyam Adil Habeeb (2018). Improving And Analysis Turbine Wheel Of Turbocharger For High-Performance Engines . *Journal of Mechanical Engineering Research and Developments*, 41(3) : 91-96.

Figure 13: Pressure contour at $\beta = 65^0$

3.1.2 Case – II Outlet Blade Angle, B = 45⁰

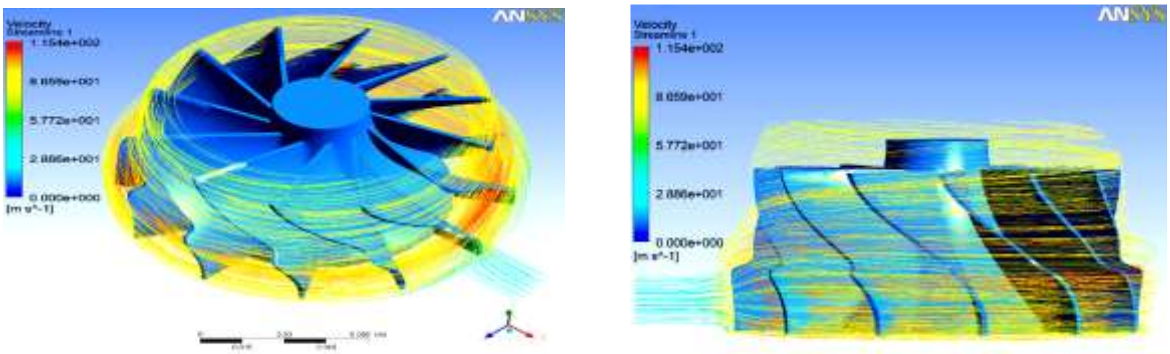


Figure 14: Velocity streamline at $\beta = 45^0$

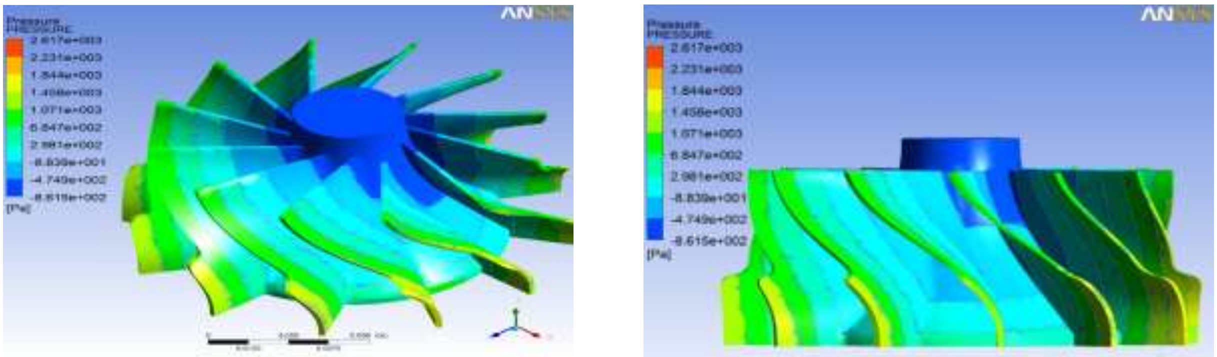


Figure 15: Pressure contour at $\beta = 45^0$

3.1.3 Case – III Outlet Blade Angle, B = 35⁰

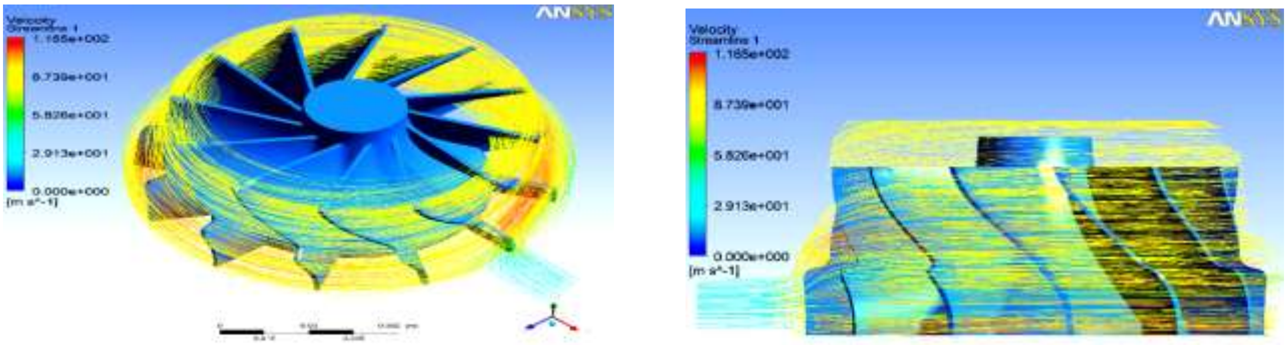


Figure 16: Velocity streamline at $\beta = 35^0$

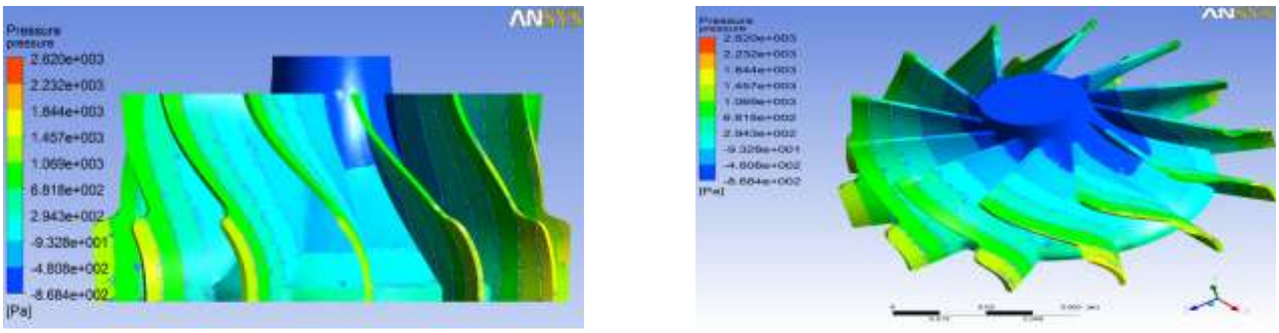


Figure 17: Pressure contour at $\beta = 35^0$

4. CONCLUSION

Depend on the above analyses for turbine wheel is the important and main component of the Turbocharger. The Details of the design and geometry specifications of the turbine wheel for a turbocharger were presented. The analysis part project is done, with the help of computational fluid dynamics (CFD) which is one of the mostly used software to analyze the fluid flow phenomena. In this study ANSYS-CFX is used for the analysis of the turbine wheel of the turbocharger to the different types of blade angles. for turbine wheel outlet blade angle, $\beta = 65^0, 45^0$ and 35^0.

The following are the conclusion drawn from the results of Computational Flow Analysis for the turbine wheel of a turbocharger:

- By observing the velocity streamline and the pressure contours. It is concluded that, for outlet blade angle, $\beta = 35^0$ is more efficient then outlet blade angles, $\beta = 65^0$ & 45^0. And outlet blade angle, $\beta = 45^0$ is more fluctuating than other blade angles.

- Therefore, it is recommended the outlet blade angle, $\beta = 35^0$ are more efficient than other blade angles.

REFERENCE

[1] Kech, J., Hegner, R., Männle, T. 2014. Turbocharging: Key technology for high-performance engines. Engine technology, 1-5.

[2] Rose, J., Winwood, S. 2015. A Novel Approach for Improving Turbine Wheel Quality Using Eddy Current Inspection. Cummins Turbo Technologies (CTT), Huddersfield, UK, 2-10.

[3] Sawadi, A.S., Shkhair, M.M., Tilefih, R.J. 2018. Optimize and Analysis Compressor Wheel of Turbo Charger.

[4] Rafi, S.M., Rao, N.A.N. 2014. Structure Analysis of a Turbocharger Compressor Wheel Using FEA. Int. Journal of Engineering Research and Applications, 4 (10), 157-159.