

Modelling and CFD Simulation of Turbocharger's Centrifugal Compressor for a Marine Diesel Engine

¹Garimella. Sai Venu Manoj, ²Dr. K. Kalyani Radha

¹M. Tech Student, ²Professor

¹Department of Mechanical Engineering, ²Department of Mechanical Engineering

¹JNTUACEA Anantapur, ²JNTUACEA Anantapur, India

Abstract: This paper describes the modelling and simulation of the centrifugal compressor of a turbocharger for a Marine Diesel Engine. The fluid domain of the centrifugal compressor comprises of inlet casing, impeller, diffuser, and volute casing. The aim is to improve the mass flow of air to the engine by changing the centrifugal compressor of a turbocharger. For this, a centrifugal compressor is modelled in CFturbo software based on the existing model. A modified model is generated from the existed model in which both the compressors require same driving power, driving torque, and are having same hub diameter. The geometry of the entire compressors is developed based on the CFturbo software. CFD simulation for the two compressors is done using Ansys FLUENT. The components of the entire compressors are meshed in the FLUENT analysis system of ANSYS 18.0 workbench. The transient state analysis has done using the K- ϵ (epsilon) turbulence model with the numerical method of pressure-based solver. Mass flow rate at the inlet and pressure outlet at the outlet are given as boundary conditions. The convergence criteria are kept at 10^{-4} . Comparison of the two compressors is made with respect to the mass flow rate and pressure at the outlet. The CFturbo results are compared with the CFD simulation result.

Keywords - Turbocharger; Centrifugal compressor; CFturbo; CFD simulation; Mass flow rate; Pressure.

I. INTRODUCTION

Turbochargers are widely used in automobiles and marine diesel engines. In ships, space and weight are of major concern. A turbocharger is a machine which increases the power output of an engine by delivering pressurized air. Turbocharger draws outside air and pressurizes it before the engine cylinder. The hot exhaust gases from the engine cylinders blow on to the turbine and make it turn at high speed driving the centrifugal compressor [1]. The centrifugal compressor is generally composed of an impeller, a diffuser, and a volute. The pressure is raised by adding kinetic energy/velocity to a continuous flow of fluid through the impeller. This kinetic energy is then converted to an increase in potential energy/static pressure by slowing the flow through a diffuser, and the fluid is gathered by a volute [2]. The increased weight of air combines with fuel to produce more power.

The main objective of this paper is to increase the mass flow of air to the engine through the turbocharger by replacing the centrifugal compressor. At certain engine's RPM, the turbocharger compressor is designed to deliver a certain mass flow of air. If the mass flow of air from the compressor increases, then more power can be acquired at lesser RPM of the engine (when sufficient quantity of fuel is injected according to the air). Generally there are two ways to increase the mass flow of air through turbocharger. (i)The turbocharger can be replaced by a different turbocharger which can give more mass flow of air. (ii)The impeller in the centrifugal compressor of a turbocharger can be changed (i.e., the impeller inlet, outlet diameters and blade angles can be changed which shows much effect on compressing the air). In this case, modification of impeller is done. For this, a turbocharger is selected (MAN TCR18 using on MAN L27/38 Marine Diesel Engine) and its centrifugal compressor design is considered [9]. As exact dimensions of the compressor are not available, the parameters (like mass flow of air, pressure ratio, speed, inlet total pressure and inlet total temperature, etc.,) were considered based on the engine and turbocharger product guides and catalogues [12][13]. The parameters were calculated at a certain turbocharger speed of 20000RPM. Depending on the values, these were given as input to the software called CFturbo which can calculate all the parameters based on the empirical relations programmed in it from which design dimensions are generated. Based on the generated design dimensions, a model has generated. The generated design dimensions are modified (inlet, outlet diameters and blade angles) and a modified compressor model has generated in CFturbo software in which both the compressors have same hub diameter, requires same driving power and torque. CFD simulation is carried out on the compressors using Ansys FLUENT. Comparison of the two compressors is made. Simulation cannot be performed in CFturbo software. Present work is aimed to analyze and simulate the pressure and mass flow rate at the outlet of the compressors.

II. DESIGN OF CENTRIFUGAL COMPRESSOR

Present work is carried out under the following assumptions; (i)The flow comes in through the inlet without any pre-swirl, (ii)The flow in the vaneless space is of a free-vortex type, (iii)The wall boundaries are having smooth surface with adiabatic flow and no slip. (iv)The volute casing is constructed of gradually increasing circular cross-sections with a constant average velocity. In order to design the centrifugal compressor; initially the design specifications are to be given as input to the CFturbo software. It will calculate the necessary parameters (geometrical and hydraulic) required for generating the CAD model of the impeller. A backswept impeller is generated depending on the blade angles. After impeller design, vaneless diffuser and volute casing with inlet are generated. The CFturbo software has an advantage that the generated values can be edited according to the user requirements by referring to the CFturbo manual which shows the optimal range for the calculated parameters [10]. Fig. 1 and Fig. 2 shows the compressors designed in CFturbo software. The backswept impeller is efficient than the radial impeller, as described in [11]. The generated compressor models are exported to Ansys SpaceClaim in the form of STP files and the necessary modifications are done to the models which are shown in Fig. 3 and Fig. 4.

2.1 Design Specification

Design of the compressor input data: Mass flow rate, total pressure ratio, Speed in RPM, Inlet total pressure, Inlet total temperature of air.

2.2 Geometric Parameter

Blade angles, number of blades, impeller discharge width, hub/tip ratio, Diameter of the inlet, Diffuser and Volute.

2.3 Hydraulic Parameter

Flow coefficient, head coefficient, blade velocity, relative velocity and other hydraulic parameters needed to describe the flow direction. (Some of the geometric and the hydraulic parameters of the impeller are applied to the inlet, stator, diffuser and volute also.)

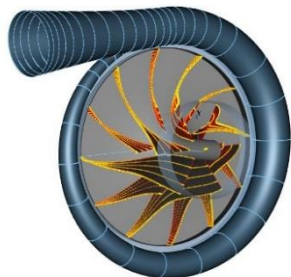


Figure 1: Centrifugal Compressor Model based on Existing design (CFturbo)

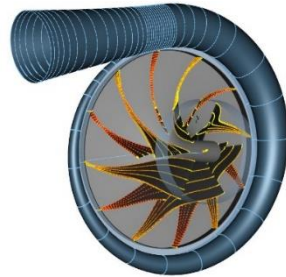


Figure 2: Modified Centrifugal Compressor Model (CFturbo)

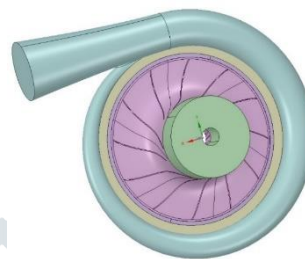


Figure 3: Existing Centrifugal Compressor Model (Ansys Space Claim)

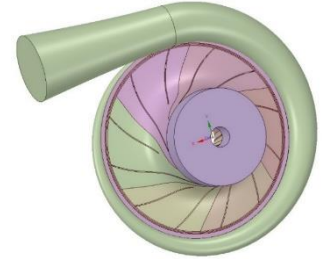


Figure 4: Modified Centrifugal Compressor Model (Ansys Space Claim)

III. MODELLING OF CENTRIFUGAL COMPRESSOR

3-D flow simulation is carried out on an axial flow centrifugal compressor using ANSYS-FLUENT analysis system. For turbomachinery, the multiple frame model and sliding mesh model are widely used to simulate the flow field between the rotating area and the stationary area [3]. In this case, the sliding mesh model is used. The CFD model of the compressor was split into six zones: inlet zone, rotating impeller zone, stator zone, diffuser zone, volute zone and outlet zone. The material of the impeller is being taken as aluminium. The solid walls such as blade surfaces, hub, and shroud are given as moving wall and others are given as stationary wall. No slip condition has imposed over all wall boundaries. The flow-field information between the rotating and stationary zones was transmitted through interfaces [14]. Ansys FLUENT is used to generate the grid of the entire centrifugal compressor.

3.1 Mesh

The main difficulty in dealing with the real geometries is achieving the appropriate mesh. In this work, a mixed polyhedral mesh was used. ANSYS-FLUENT makes it possible to generate a polyhedral mesh from a tetrahedral mesh using the algorithm described in [5]. The greatest advantage of polyhedral mesh is that they are more accurate than the equivalent tetrahedral mesh. With fewer cells also the same resolution can be achieved, as described in [6]. However, the relative error decreases as the number of cells in the mesh increases, described in [3]. Polyhedral mesh has been used to solve computational problems in engineering. A method for solving Reynolds equations with polyhedral cells, showed better behaviour than quadrilateral cells in certain problems, as in [7]. A polyhedral mesh for the turbocharger computations is used, cited less numerical diffusion and a more accurate gradient calculation as the benefits of this type of mesh, as in [8]. As shown in the Fig. 5 and Fig. 6; Fig. 7 and Fig. 8, hexahedral mesh was adopted initially and then converted into polyhedral to get optimum orthogonal quality and aspect ratio respectively. Table 1 shows

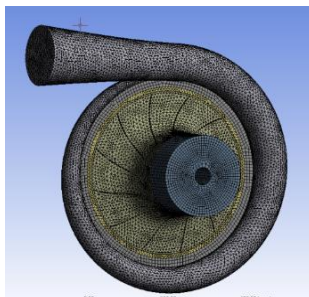


Figure 5: Existing Centrifugal Compressor Model (Fluent Mesh)

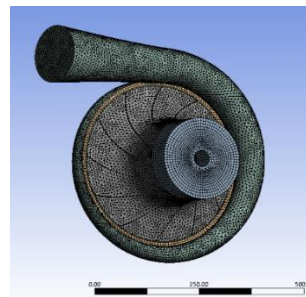


Figure 6: Modified Centrifugal Compressor Model (Fluent Mesh)

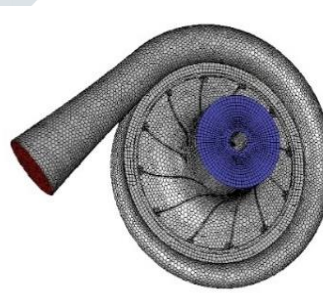


Figure 7: Existing Centrifugal Compressor Model (CFD Mesh)

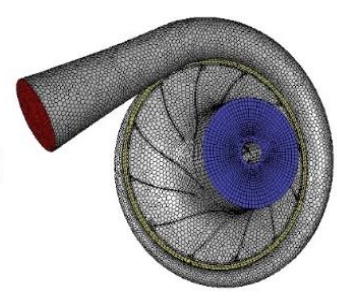


Figure 8: Modified Centrifugal Compressor Model (CFD Mesh)

the mesh statistics for an existing model and modified model of a centrifugal compressor.

Table 1: Mesh Statistics of a centrifugal Compressor for an Existed Model and a Modified Model

Parameters	Mesh Statistics	
	Existing Model	Modified Model
Nodes	419081	365998
Elements	93152	84071

3.2 Boundary Conditions

The CFD simulations were carried out with the commercial code Fluent® from ANSYS® [4]. The transient simulations were periodic, when the monitored inlet and outlet mass flow rate were kept constant and the residuals were under 10^{-4} . In this case, the transient state analysis is applied using K-ε (epsilon) turbulence model with pressure-based solver for the flow simulation of the centrifugal compressor. The Boundary conditions used for the flow simulation are summarized in Table 2.

Table 2: Boundary Conditions

Boundary Conditions	
Parameters	Boundary Conditions
Solver	Pressure-based
Fluid	Air
Inlet	Mass flow inlet
Outlet	Pressure outlet
Wall	No slip
Turbulence model	Realizable K-ε [Epsilon]
Solution Methods	Coupled
Transient Formulation	Second Order Implicit
Maximum residual convergence criteria	10^{-4}

IV. RESULTS AND DISCUSSION

Fig. 9[a, b, c, d] shows the pressure distribution and velocity magnitude for an existing centrifugal compressor model. Fig. 10[a, b, c, d] shows the pressure distribution and velocity magnitude in a modified centrifugal compressor model. The comparison of the two compressors is shown in table 3. The table 4 and table 5 shows the comparison between the CFturbo and Ansys Fluent values for an Existing and Modified models respectively.

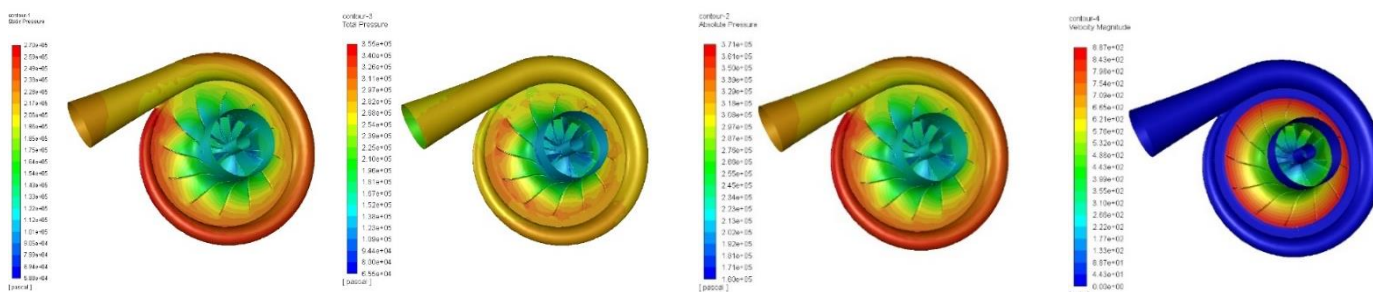


Figure 9-a: Static pressure distribution

Figure 9-b: Total pressure distribution

Figure 9-c: Absolute pressure distribution

Figure 9-d: Velocity distribution

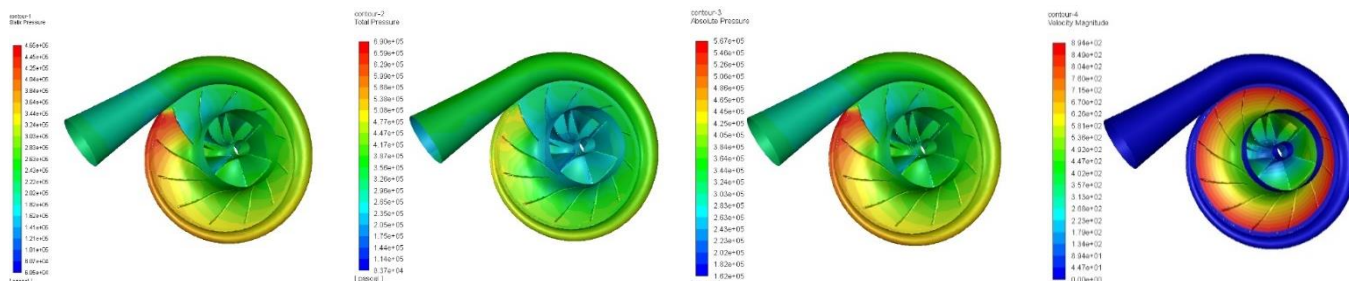


Figure 10-a: Static pressure distribution

Figure 10-b: Total pressure distribution

Figure 10-c: Absolute pressure distribution

Figure 10-d: Velocity distribution

Table 3: Comparison of the Existing and Modified Models

Comparison of the Existed and Modified Models			
Existing Model		Modified Model	
Parameters	Values	Parameters	Values
Impeller Diameter at Inlet [mm]	218.7	Impeller Diameter at Inlet [mm]	223.2
Impeller Diameter at Outlet [mm]	442.3	Impeller Diameter at Outlet [mm]	423.5
Blade angles at Inlet [°]	$\alpha_1=90$ $\beta_1=30.1$	Blade angles at Inlet [°]	$\alpha_1=90$ $\beta_1=33.3$
Blade angles at Outlet [°]	$\alpha_2=14$ $\beta_2=30.5$	Blade angles at Outlet [°]	$\alpha_2=17.8$ $\beta_2=30.8$
Required Driving Power [kW]	538	Required Driving Power [kW]	538
Required Driving Torque [Nm]	256	Required Driving Torque [Nm]	256
Hub Diameter [mm]	46.47	Hub Diameter [mm]	46.47
Static Pressure at Outlet [bar]	2.22	Static Pressure at Outlet [bar]	2.06
Absolute Pressure at Outlet [bar]	3.23	Absolute Pressure at Outlet [bar]	3.08
Mass flow rate at Outlet [kg/s]	3.36	Mass flow rate at Outlet [kg/s]	3.96
Volume flow rate at Outlet [m ³ /s]	2.74	Volume flow rate at Outlet [m ³ /s]	3.23

Table 4: Comparison of the CFturbo and Ansys Fluent values for an Existing Model

Comparison of the CFturbo and Ansys Fluent			
CFturbo		Ansys Fluent	
Parameters	Values	Parameters	Values
Static Pressure at Outlet [bar]	2.22	Static Pressure at Outlet [bar]	2.22
Absolute Pressure at Outlet [bar]	3.54	Absolute Pressure at Outlet [bar]	3.33
Mass flow rate at Outlet [kg/s]	3.36	Mass flow rate at Outlet [kg/s]	3.36
Volume flow rate at Outlet [m ³ /s]	2.83	Volume flow rate at Outlet [m ³ /s]	2.74

Table 5: Comparison of the CFturbo and Ansys Fluent values for a Modified Model

Comparison of the CFturbo and Ansys Fluent			
CFturbo		Ansys Fluent	
Parameters	Values	Parameters	Values
Static Pressure at Outlet [bar]	2.06	Static Pressure at Outlet [bar]	2.06
Absolute Pressure at Outlet [bar]	3.07	Absolute Pressure at Outlet [bar]	3.08
Mass flow rate at Outlet [kg/s]	3.96	Mass flow rate at Outlet [kg/s]	3.96
Volume flow rate at Outlet [m^3/s]	3.34	Volume flow rate at Outlet [m^3/s]	3.23

V. CONCLUSION

This paper describes the CFD simulation of a Turbocharger's centrifugal compressor. A model of the centrifugal compressor has been generated based on the existing design and a modified model is generated using CFturbo software. The models generated in CFturbo software is exported to Ansys SpaceClaim in the STP file format. The air mass flow rate and the pressure distribution are investigated by using the Ansys-Fluent analysis system. The two compressors are compared and the mass flow rate of air from the modified compressor model is more than the existing compressor model. Both the compressors are having the same hub diameter [46.47mm] and require the same driving power [538kW] and torque [256Nm]. The modified model produces increased mass flow rate at slightly lower pressure at outlet than the existing model at a constant speed of 20000 RPM as shown in table 6. Fig 11. Shows the performance of centrifugal compressors when rotating at 20000RPM in which surge criteria for the modified compressor is improved.

Fig. 12 and Fig. 13 shows the static and absolute pressure at outlet of the existing model and modified model respectively. The pressure distribution and the air mass flow rate depends on the speed, inlet-outlet diameters and blade angles of the impeller. Similar computational simulation models can also be used for analyzing the pressure, velocity and stress distribution of the turbines, pump, fan and blower.

Table 6: Simulation Results Comparison for the Existing Model and the Modified Model

Simulation Results Comparison			
Existing Model		Modified Model	
Parameters	Values	Parameters	Values
Static Pressure at Outlet [bar]	2.22	Static Pressure at Outlet [bar]	2.06
Absolute Pressure at Outlet [bar]	3.23	Absolute Pressure at Outlet [bar]	3.08
Mass flow rate at Outlet [kg/s]	3.36	Mass flow rate at Outlet [kg/s]	3.96

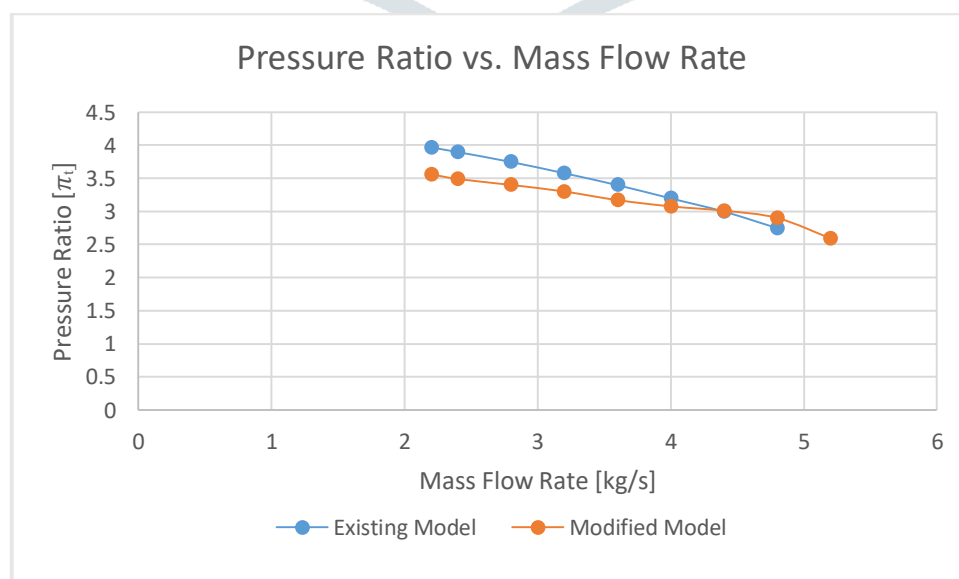


Figure 11: Pressure Ratio vs. Mass Flow Rate for an Existing and Modified model [20000RPM]

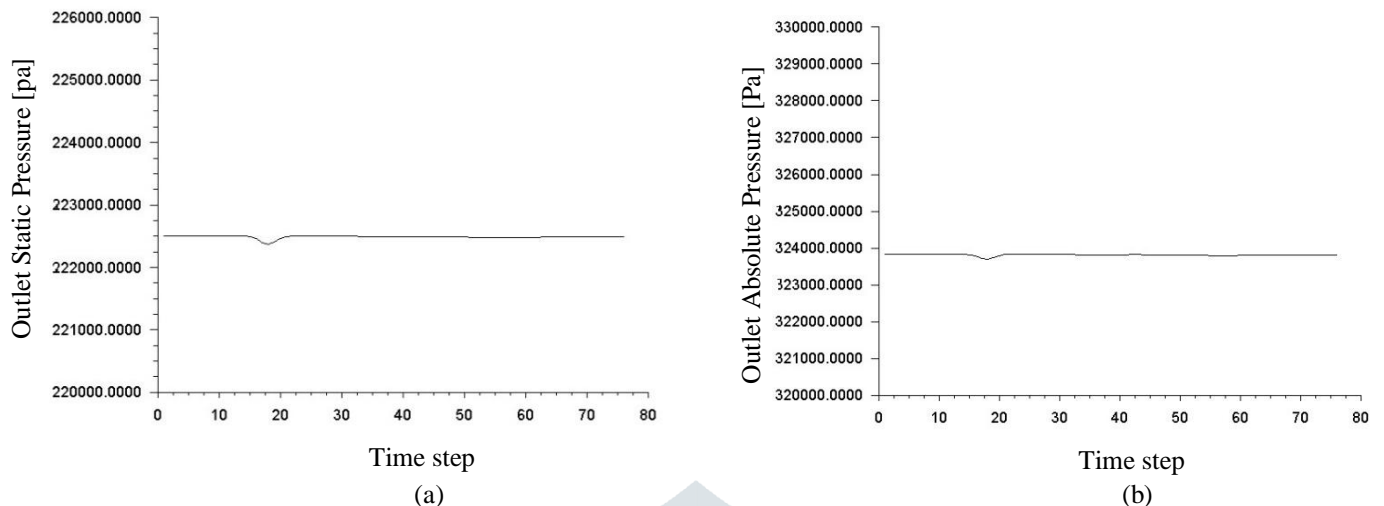


Figure 12: Pressures at the Outlet of the Volute for an Existing Model
(a) Static Pressure vs. Time step (b) Absolute Pressure vs. Time step

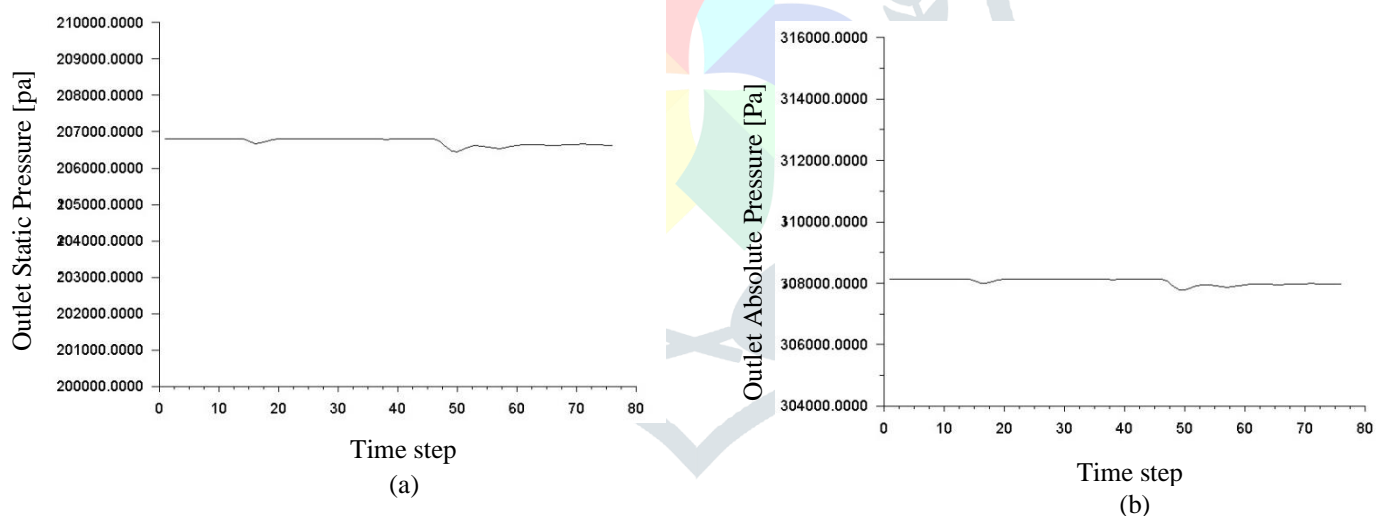


Figure 13: Pressures at the Outlet of the Volute for a Modified Model
(a) Static Pressure vs. Time step (b) Absolute Pressure vs. Time step

VI. ACKNOWLEDGEMENT

I express my deep sense of gratitude to my guide Dr. K. Kalyani Radha, Professor, Department of Mechanical Engineering, JNTUACEA, Anantapur for their incessant support, motivation and guidance until the completion of my dissertation and journal. I am thankful for their benevolence and consideration, at every stage of this endeavour and many others.

REFERENCES

- [1] What are Turbochargers? Explaining how a turbocharger works?
<https://www.melett.com/technical/what-are-turbochargers/>
- [2] Kaijun Wei, Shuguang Zuo, Huijuan He, and Zhe Wang, "Numerical study on the unsteady behaviour of a centrifugal compressor for the fuel-cell vehicle," Acoustical Society of America. <https://doi.org/10.1121/2.000032>
- [3] J. Galindo, S. Hoyas, P. Fajardo & R. Navarro (2013), "Set-Up Analysis and Optimization of CFD Simulations for Radial Turbines," Engineering Applications of Computational Fluid Mechanics, 7:4, 441-460, DOI: 10.1080/19942060.2013.11015484.
- [4] ANSYS, "Module: 10 Transient flow modelling". © 2016 ANSYS, Inc. April 12, 2016
- [5] ANSYS (2009), "Ansys Fluent 12.0 User's Guide," Canonsburg, PA: ANSYS Inc.

- [6] FLUENT (2006), "FLUENT 6.3 Validation Guide".
- [7] Trithart M, Gutknecht D (2007), "Three-dimensional simulation of free-surface flows using polyhedral finite volumes," Engineering Applications of Computational Fluid Mechanics 1(1):1-14.
- [8] Baris O, Mendonça F (2011), "Automotive turbocharger compressor CFD and extension towards incorporating installation effects," Proceedings of the ASME Turbo Expo 2011: Power for Land, Sea and Air, GT2011-46796.
- [9] MAN TCR Turbocharger-The Cutting Edge, "MAN Diesel & Turbo," 86224 Augsburg, Germany.
- [10] CFturbo GmbH, "CFturbo help topics," CFturbo Version 10.3, 2017.
- [11] Schwind and Abdallah, Global J Technol Optim 2015, 6:3, "A Look at Compressor Impeller Technologies for Turbochargers Focusing on Surge Mitigation". DOI: 10.4172/2229-8711.1000185.
- [12] Project Guide, TCR Turbocharger, EN-US Turbocharger Copyright © 2014, "MAN Diesel & Turbo MAN Diesel & Turbo," 86224 Augsburg, Germany.
- [13] MAN L27/38 PROPULSION Catalogue "MAN Diesel & Turbo MAN Diesel & Turbo," 86224 Augsburg, Germany
- [14] Have a nice time! (2017, November 7). A centrifugal fan simulation in Ansys Fluent sliding mesh, periodic interfaces Blade Gen Fluent, FFT [Video file]. Retrieved from https://youtu.be/eD0nxctN_yw.

