

GT2017-64359

DEVELOPMENT OF EFFICIENT COMPRESSORS FOR TURBOCHARGERS

Dhinakaran Ramachandran
 Turbo Energy Private Limited,
 Chennai, India
 dhinakaran.r@turboenergy.co.in

Balamurugan Mayandi
 Turbo Energy Private Limited,
 Chennai, India
 balamurugan.m@turboenergy.co.in

Seran Krishnamoorthy
 Turbo Energy Private Limited,
 Chennai, India
 seran.k@turboenergy.co.in

Gopalakrishnan Mani
 Turbo Energy Private Limited,
 Chennai, India
 gopalakrishnan.m@turboenergy.co.in

Vasudevan Ramesh
 Turbo Energy Private Limited,
 Chennai, India
 vasudevan.r@turboenergy.co.in

Swathi Chitra Lakshmanan
 Turbo Energy Private Limited,
 Chennai, India

ABSTRACT

Automotive turbochargers play an important role in improving fuel economy, reducing emissions and improving drivability. Key to the improvement of the turbocharger performance is compressor efficiency. Compressors used in turbochargers are typically operated in a wide range of speed and flow. This wide operating range is a challenge to the design and improving the performance is often a fine balance between required efficiencies towards the surge, choke regions apart from having a comfortable speed margin for high altitude operations. In this study an existing compressor that best matched a 180hp commercial diesel engine application is chosen and its performance is further improved towards the lower flow region. Improvement is carried out through a set of designed experiments using a combination of Preliminary Design (PD) and Computational Fluid Dynamics (CFD) tools. Mechanical integrity of the wheel is ensured using Finite Element Analysis. A prototype is made out of the improved design and tested in an in-house gas stand. Predicted efficiency improvements are reflected in gas stand tests. Efficiency improvements in the lower flow range are observed over 7% while there is an acceptable

drop (3.7%) near the peak power side. The improved compressor also shows higher part load efficiencies.

NOMENCLATURE

b2	Exduser width
BL	Blade axial length
BSIV, VI	Bharath Stage IV, VI
CFD	Computational Fluid Dynamics
DoE	Design of Experiments
D1h	Inlet wheel diameter at hub
D1t	Inlet wheel diameter at tip, Inducer diameter
D2h	Outlet wheel diameter at hub
N	Design shaft speed
PD	Preliminary Design
SST	Shear Stress Transport
y ⁺	Dimensionless wall distance
Z	Number of blade set

Greek

β1	Mean blade angle at inlet
β2	Mean blade angle at outlet

INTRODUCTION

More and more government norms for emissions are getting stringent. The Indian government is ambitiously pushing vehicle emission limits from BSIV to BSIV (equivalent to Euro IV & VI) by year 2020. Turbochargers play an important role in reducing emissions, improving fuel economy and performance [1]. In a typical turbocharger a radial turbine is connected to a radial compressor which compresses the air before sending to the engine air intake system. Compressor efficiency contributes to overall turbocharger efficiency and more importantly tailoring the characteristics of the compressor is a key to improved performance in certain parts of the engine operation [2]. In certain markets where the vehicles operate at lower speeds and often under idling conditions the engine and the compressor tends to operate in a low flow regime often. Therefore, the compressor needs to be efficient in this region, although one has to ensure that the compressor has high flow capability. This contrasting design requirements is quite challenging and often calls for carefully tailored compressors. In general, this requirement also stems from a need to improve the transient response of the turbocharger and the engine. Figure 1 shows a typical compressor map (all data are normalized with reference to certain reference values which are kept constant throughout the paper) with the engine operating line overlaid. The focus for this study is to improve the compressor efficiency in the points A & B without sacrificing significantly on points C & D. Point D corresponds to peak power and the engine will operate at this point rarely and hence some sacrifice here would be acceptable.

Methods for the design and analysis of performance of the turbomachines are found in detail in ref. [3-5]. Effect of individual blade parameters such as blade trim, back sweep, meridional contour on performance are presented in ref. [6-8]. Tamaki et al. [9] presented a design study to increase stage pressure ratio which includes double splitter. An approach to improve performance is to improve the overall efficiency of the compressor. This has merits in applications where the operating line falls along the locus of maximum efficiency. However, for an automotive turbocharger one needs to consider an operating line which does not follow the maximum efficiency curves. In this situation the improvement has to be made in the low flow regions without compromising the high flow performance. Chen et al. [10] describe a Design of Experiment (DoE) methodology for multi-point design of a turbocharger compressor for a gasoline engine. A combination of DoE and Three-Dimensional CFD approach is used to improve a compressor performance for gasoline engine applications. Demeulenaere et al. [11] demonstrated the use of a genetic algorithm based multi-disciplinary multi-point optimization of a compressor wheel. The study involved analysis of 180 different compressor geometries at three different operating conditions. The time taken for this exercise was a cumulative CPU time of 28 days with 4 CFD solvers running on 64 cores. Multi-disciplinary optimization of compressors has been presented recently by

Perrone et al. [12] for a centrifugal compressor and by Diener et al. [13] for a mixed flow compressor. These studies involved a large number of calculations. An approach that takes less time consuming would be quite useful considering the short design cycle time in automotive industry. The motivation for this study was primarily to develop such a methodology that will help develop compressor wheels faster to meet customer demands.

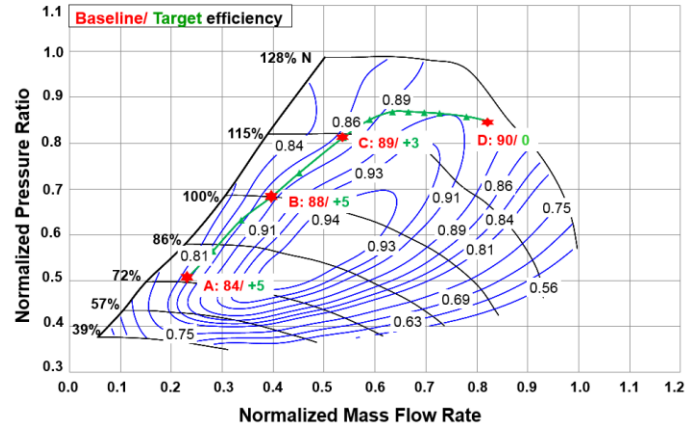


Fig. 1: Typical turbocharger map and engine operating line (Points A, B, C, D are the focus).

APPROACH

In this study a more pragmatic DoE approach along the lines of Chen et al. [10] is used to improve compressor performance in the low flow rate regime. A baseline compressor is chosen and is numerically modelled using a PD tool AxSTREAM [14] as well as a full Three-Dimensional CFD solver ANSYS-CFX [15]. First the PD tool is calibrated with the test data for the baseline compressor, which typically involves a small change to the loss coefficients [16]. Then the CFD solver is calibrated to predict compressor performance. The SST model with Sapart & Shur correction [17] was found the best to match pressure ratio and efficiency at higher rotational speeds.

After establishing acceptable match with gas stand test data the simulation tools are used to understand the influence of various parameters that impact the performance of the compressor. The performance in terms of pressure ratio and efficiency are evaluated at various points along the engine operating line. Geometric variables that impact the compressor performance are identified and an eight factorial DoE has been constructed. A set of 24 experiments constituting a fractional factorial experiment [18] is designed and conducted using the PD tool. Similar exercise using the PD tool has been performed by Moroz et al. [19]. The results of the simulation experiments are viewed in terms of main effects and interaction between the variables. Based on these experiments the critical factors influencing low speed/ low flow performance are identified and further improved to obtain a compressor with improved low-end efficiency.



Fig. 2: Turbocharger performance test facility.

A few additional design iterations are performed using the variables identified through the DoE. The performance of the improved wheel is evaluated using the CFD solver. Physical prototype of the improved compressor was then made and its performance tested in the in-house gas stand.

TEST FACILITY

The in-house test facility for compressor and turbine performance built with the assistance of BorgWarner Turbo Systems [20] has been used for measuring the compressor performance. The gas stand uses a natural gas/ propane burner with mass flow capacity of 0.29 to 0.75 kg/s, with a peak burner output of 630 kW. This has the ability to test typical compressors used in commercial diesel engine applications. The turbocharger mounted in the gas-stand is shown in Fig. 2. In the experiments surge is first detected through sensors which sense the variations in compressor inlet pressure, temperature, mass flow fluctuations as well as compressor outlet pressure and temperature. Likewise choke flow is determined and once the surge and choke limits are identified the intermediate points are filled in automatically in the tests. The uncertainty of the various instruments used are within 1%.

NUMERICAL MODELLING

The baseline compressor information such as the blade geometry are imported into the PD tool. The baseline compressor also included inlet flow recirculation port and this was also modelled in the PD tool. The baseline compressor model is shown in Fig. 3. Once the basic compressor design parameters (such as b_2 , BL , $D1h$, $D1t$, $D2h$, N , β_1 , β_2 , Z) are provided as input the PD tool solves simplified flow equations [19] in order to get a complete compressor performance map in a few minutes. The relatively high speed with which one can get a full

performance map at reasonable accuracy lends PD tool for DoE at an initial screening level. The choice of the geometric parameters is from Japikse [4] and the ease of drop-fitting a new wheel in to the baseline compressor housing.

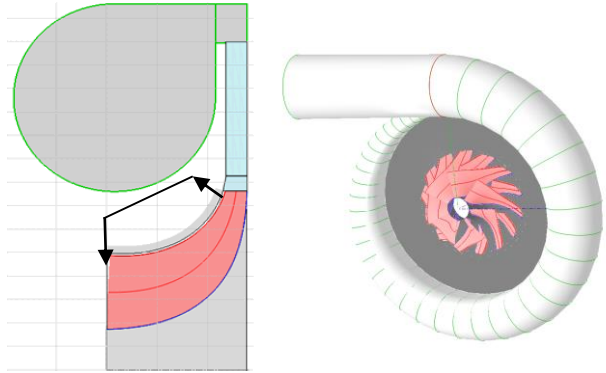


Fig. 3: Baseline compressor model in the preliminary design tool.

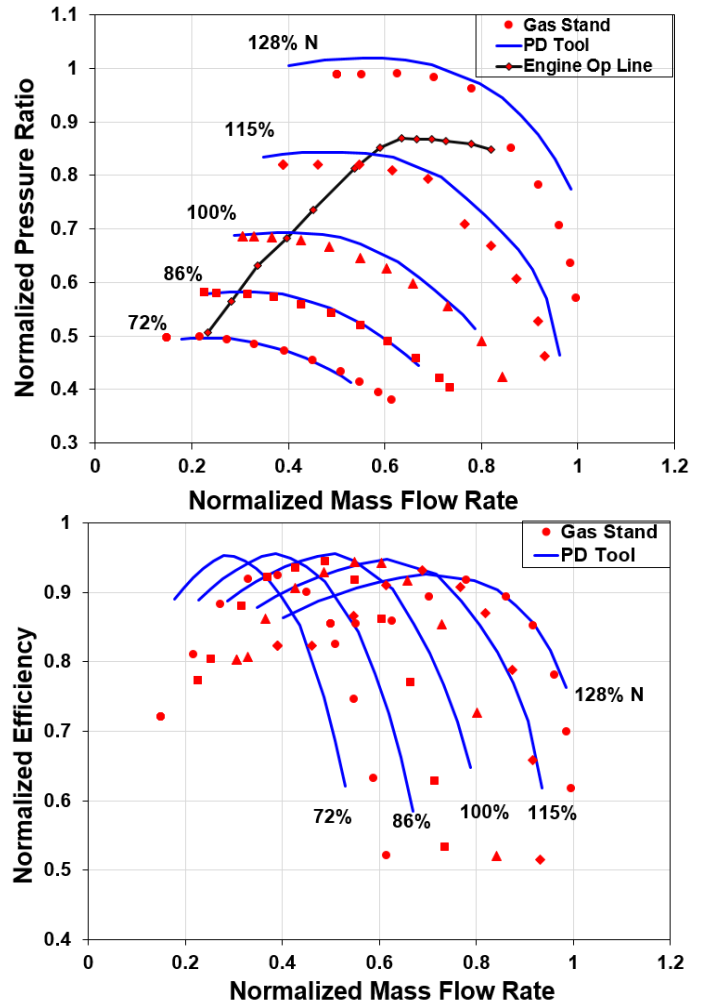


Fig. 4: Validation of preliminary design tool.

The predicted performance of the baseline compressor for the specified geometric and operating conditions is compared with the test results in terms of pressure ratio and efficiency (Fig. 4) at different rotor speeds. Predictions of pressure ratio matches well with the test data across all speeds. At lower flow rates test data shows lower efficiency compared to predictions. Overall, with the accuracy shown, the PD tool can be expected to capture the qualitative impact of design changes.

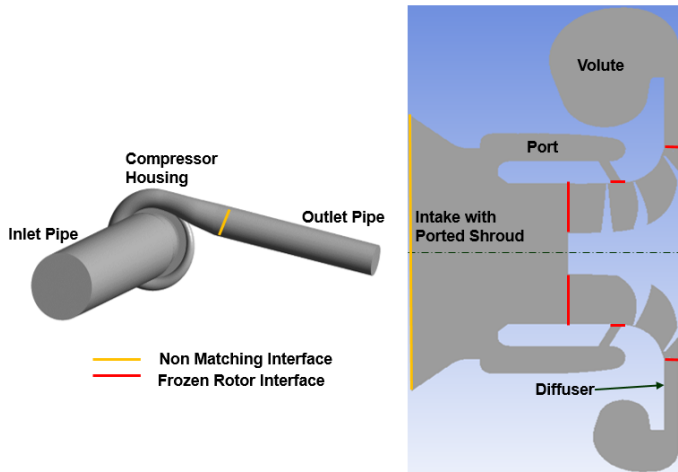


Fig. 5: Schematic of the computational domain.

Detailed CFD modelling of the baseline compressor is performed in the CFD solver [15]. The computational domain inlet is taken at five times the diameter of the intake (which has a ported shroud) and ends after the volute exit at a distance five times the diameter of volute exit (Fig. 5). Interfaces between the rotating and stationary components are modelled using the “Frozen-Rotor” approach, which is generally recommended for centrifugal machines where the exit from the rotor sees an asymmetric geometry downstream. This approach is however more computationally expensive compared to the alternative “mixing-plane” approach where only one impeller blade passage needs to be modelled. The locations of interfaces are indicated in Fig. 5. The computational domain comprised of inlet pipe, intake, impeller, diffuser and volute is meshed using different types of grid. Structured mesh is used except in the diffuser and volute where unstructured tetrahedral mesh with prismatic layers near the walls are used. The grid near the wall is fine enough to (y^+ values < 30) be used with the SST turbulence model with automatic wall function approach [15]. Typical boundary conditions are total pressure, total temperature, flow angle (assumed axial) at domain inlet and static pressure at the domain outlet. The SST turbulence model is used for simulating fine scale turbulence, which is more conservative in the predictions of efficiency. Since a numerical model depends somewhat on the grid size used it is important to understand and quantify its effects. Three different grids (1.5Million, 4 Million and 16 Million) were used and the performance in terms of pressure ratio and efficiency are obtained for a typical operating condition. Difference in pressure ratio was less than 0.1% (not

shown) and efficiency less than 0.5% (Fig. 7) between the medium and fine grid results and hence the medium grid of 4Million was used for further simulations. This is comparable to the mesh size of 2Million cells used in ref. [12].



Fig. 6: Computational grid (inlet and outlet extensions not shown).

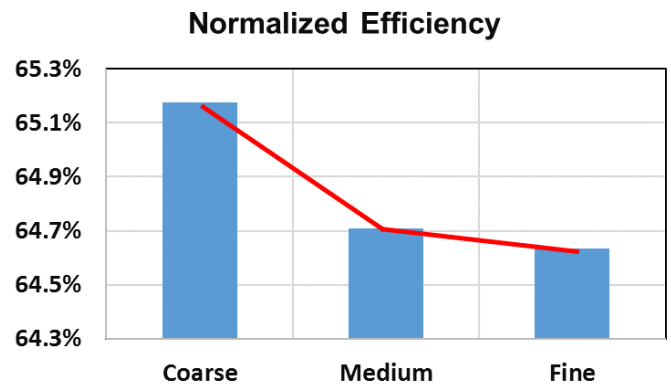


Fig. 7: Sensitivity of performance to mesh refinement.

The predicted performance of baseline compressor is compared with the gas stand test data in terms of pressure ratio and efficiency for three speeds (Fig. 8). Predicted pressure ratio matches closely with the test data within 1.7% in the range shown. However, the experimental operating range towards the lower flow side (towards surge) is more. This is expected since beyond a certain point the flow is essentially unsteady and it is difficult to converge the CFD calculations with steady state approximations. This implies that the CFD model used is more conservative in predicting surge line. The efficiency predictions from CFD are matching with the test data except near the lower speed. At the lower speed it is expected that the heat transfer from the turbine side is influencing the compressor efficiency

[21-23]. Both the PD tool predictions as well as the CFD predictions are higher towards the lower speeds as in simulations heat transfer effects are neglected. Nevertheless, since the interest here is to predict the impact of design changes on performance the predictive capability of the CFD tool is considered adequate. The final test of the method will be the comparison with the test data.

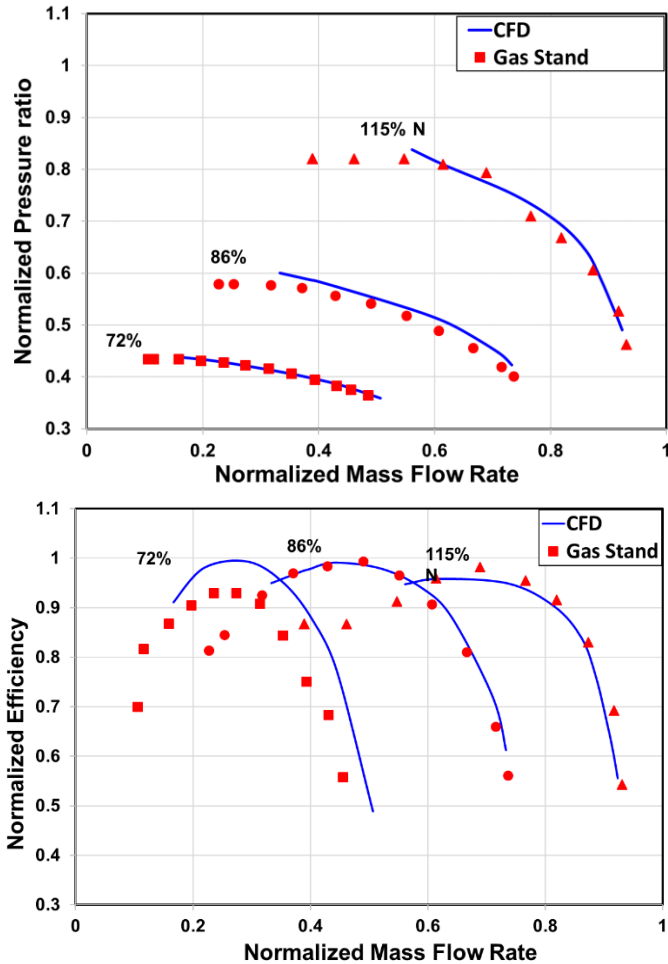


Fig. 8: Validation of the CFD tool used.

Apart from the quantitative predictions it is of interest to verify the results from simulations in terms of flow features that are captured. A typical flow feature that is of interest is the flow in the ported shroud. At lower flow rates and speeds near 100% N, the port is expected to pass a certain part of the flow back into the inlet and at higher flow rates the flow from the inlet is split and a certain part of the flow goes through the port in to the impeller. This results in an enlarged map for the compressor with ported shroud (both at the surge and choke side) for this speed. This phenomenon is clearly captured in the computed flow field at operating conditions towards surge and choke as shown in Fig. 9. Thus the simulations are able to capture qualitatively expected trends.

DESIGN OF EXPERIMENTS

An eight factor two level DoE is constructed out of seven geometric parameters shown in Fig. 10. Blade number is added as the eighth factor. These factors are varied typically 5% below and above the mean values. A full factorial DoE for eight factors would result in 256 experiments and therefore a fractional factorial experiment is setup [24] that results in 24 experiments. For obtaining the key parameters that influence the design an initial DoE with factorial experiment should suffice.

For each of the 24 experiments conducted a compressor geometry is obtained and this model is used in the PD tool. A compressor map is obtained for each of the models and performance data in terms of pressure ratio and efficiency are obtained for the four different operating points (A, B, C & D) shown in Fig. 1.

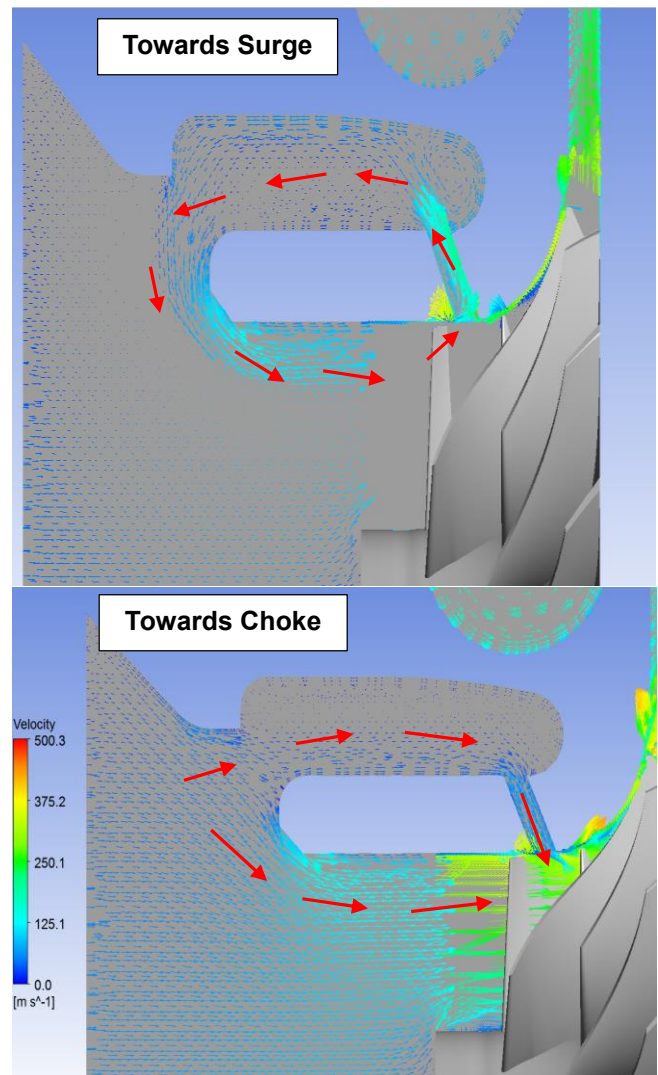


Fig. 9: Typical flow field near surge & choke.

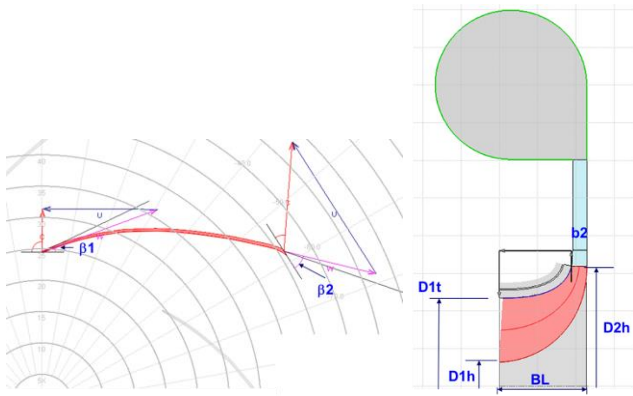


Fig. 10: DoE parameters.

The results are analyzed graphically for the four different operating points. Figure 11 shows the main effects chart for efficiency at lower and higher speeds (points A & D). For lower speed there are three major factors that influence the efficiency namely the wheel diameter at hub (D2h), outlet width (b2) and blade back sweep (β_2). The other factors that influence but lower in degree are blade axial length (BL), inducer diameter (D1t) and blade number (Z). At the higher speed for point D, there are two major factors namely inducer diameter (D1t) and wheel diameter at hub (D1h). The direction of influence of these parameters is indicated in the main effects plot (Fig. 11). Three out of the eight factors show inverse influence i.e. for example decreasing inducer diameter (D1t) helps in lower speed (point A) while it decreases the efficiency at higher speed (point D). This is understandable since a lower D1t will increase relative Mach number at inlet and will therefore increase shock losses quite rapidly near higher speeds. However, a lower D1t will mean a relatively higher velocity near lower flow range and hence beneficial. Decreasing β_2 is beneficial for lower flow without adversely affecting the efficiency near higher flow. Therefore, factors such as D1t, β_1 and β_2 needs to be carefully adjusted based on the region where improvement is sought.

IMPROVED DESIGN

An improved design is obtained by setting the design parameters based on the DoE results (Fig. 11) as well as with a few additional iterations. For each of these iterations a compressor map is obtained in the PD tool and the performance of the improved design is compared with that of the baseline design. Table 1 shows the major changes in the design parameters from the baseline design. Two changes are made that are opposite to the directions indicated by the DoE. The inlet hub diameter was increased slightly from a structural design perspective. The small increase in wheel outlet diameter at hub (D2h) was needed to maintain the pressure ratio similar to that of the baseline wheel.

Figure 12 compares the performance of the improved design with that of the baseline. The improved design shows higher efficiencies for the lower flow range while at the higher flow side it is nearly the same in terms of efficiency.

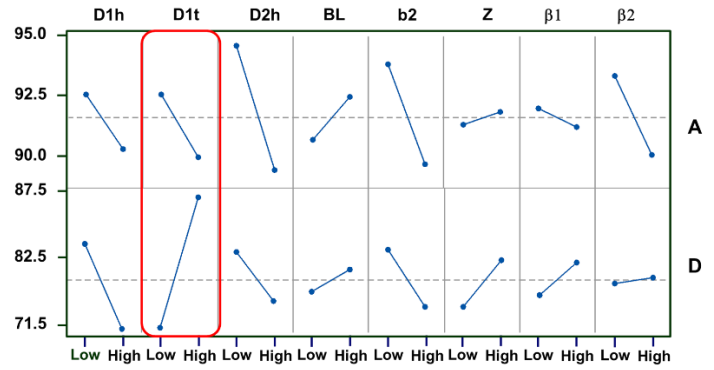


Fig. 11: Impact of geometric parameters on efficiency at points A & D.

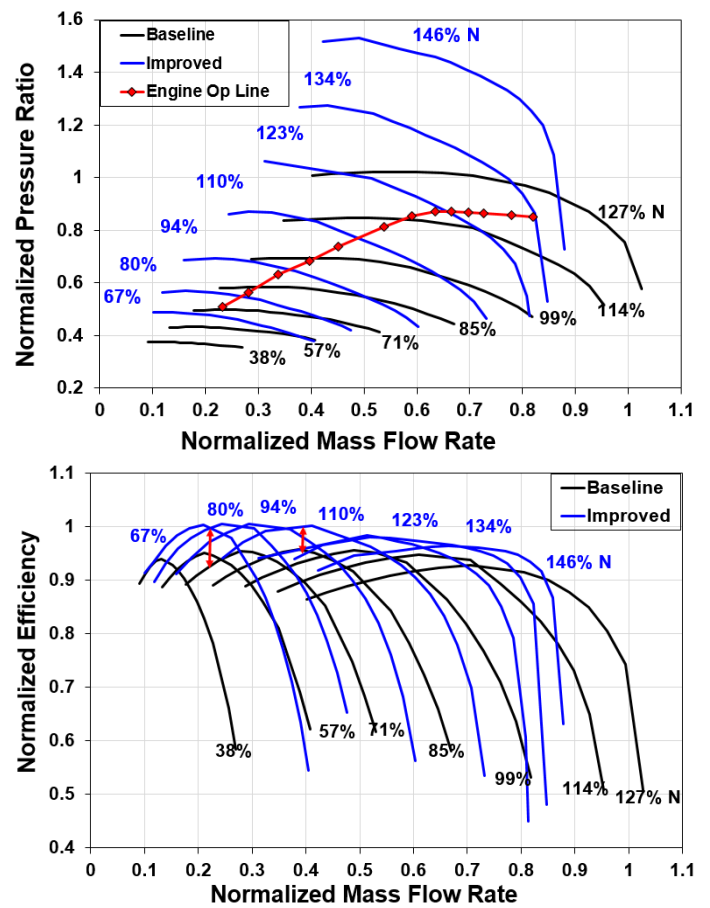


Fig. 12: Comparison of baseline and improved compressor performance in preliminary design tool.

Tab. 1: Major changes in the improved design.

Parameter	Change in Improved Design
D1h	5%
D1t	-8%
D2h	2%
BL	2%
b2	-0.3%
Z	1
β_1	3.3°
β_2	-19.6°

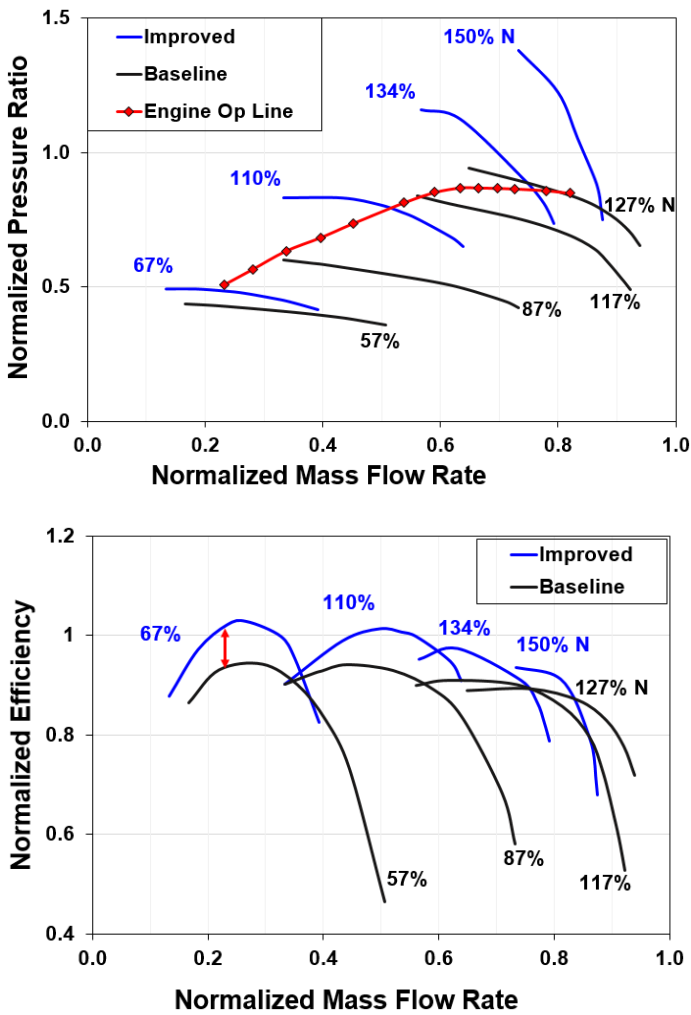


Fig. 13: Comparison of baseline and improved compressor performance in CFD tool.

Three Dimensional CFD simulations are carried out for the improved design for a few speeds and the improvement in terms of the low-end efficiency is evident (Fig. 13). Structural design of the improved design is carried out using Finite Element Analysis in Ansys [15]. The structural design criteria in terms of maximum stress and vibration limits are met by adjusting the blade thickness and fillet radii (Fig. 14). A physical prototype of the improved design is made in the in-house 5-axis milling machine, using a combination of flank and point milling.

The improved compressor performance map along with the targeted engine operating line is shown in Fig. 15. The efficiency from baseline map (Fig. 1) and CFD (Fig. 13) on the engine operating points are also marked alongside the ones from the improved compressor test map. The increase in (normalized) efficiency between the baseline and the improved compressor is 9.4% at point A, is 7.6% at point B, is 5% at point C. There is however a drop of 3.7% at the high power setting (Point D). Since low flow efficiency (at points A & B) is more important and since it has improved by more than 7% this design exercise can be considered a success. A detailed view of the flow field (Fig. 16) at low speed near surge shows a reduction in the recirculation zone present in the baseline compressor. The baseline compressor also shows a higher entropy in the low flow regime (Fig. 17) compared to the improved design. The improved design has overall lower blade loading (Fig. 18), which contributes to the higher efficiency.

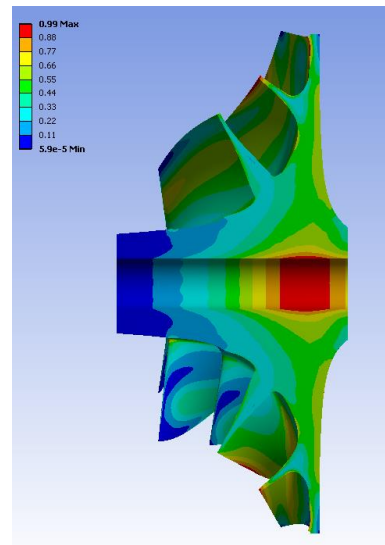


Fig. 14: Normalized stress distribution.

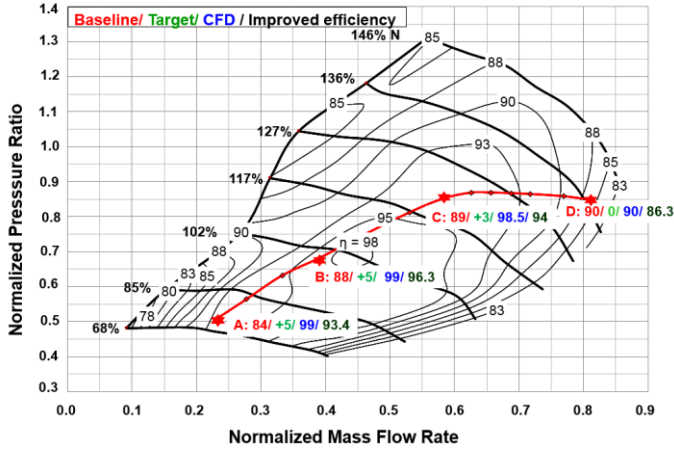


Fig. 15: Compressor map for improved design.

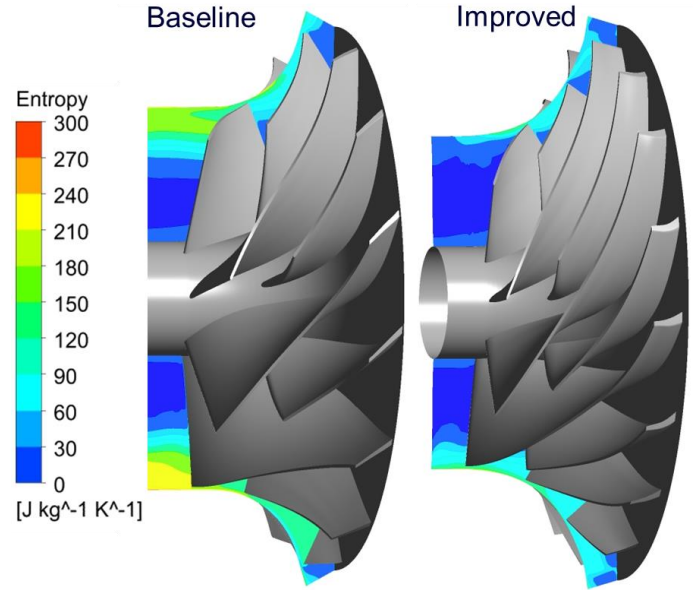


Fig. 17: Comparison of entropy in the low flow region.

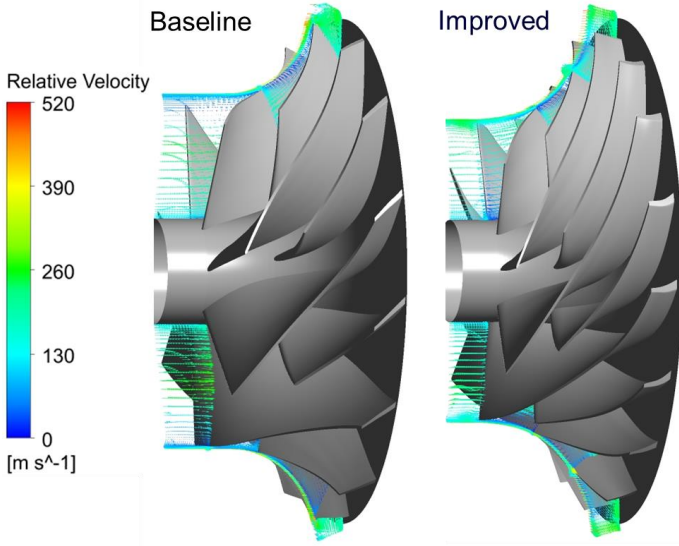


Fig. 16: Comparison of flow field in the low flow region.

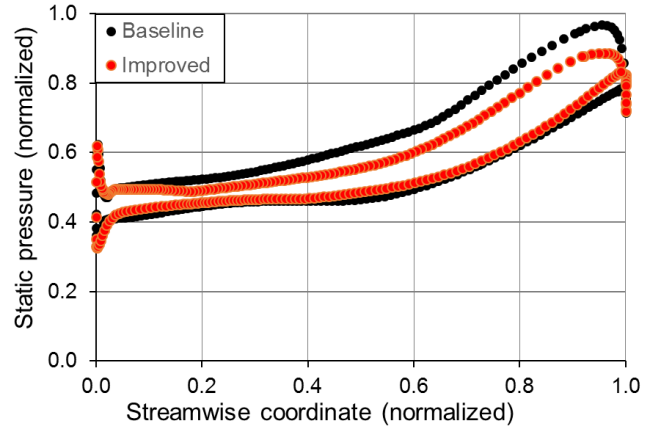


Fig. 18: Blade loading at 70% span.

CONCLUSIONS

A baseline turbo charger compressor for a 180 hp engine has been improved upon using a combination of PD and full three-dimensional CFD tools along with a DoE approach. Improvement target was in the low flow region which is of paramount importance to markets such as India where the vehicle drive cycle is such that the engine operates in idling and part load often. The study demonstrates a viable approach to designing such compressors that will have improved low end efficiencies by more than 7%. More importantly a quick approach to improve compressor performance in a turbocharger for an automotive application through numerical modelling and testing is laid out.

ACKNOWLEDGEMENT

The authors gratefully acknowledge the support for this study by the senior management of Turbo Energy Pvt. Ltd. and Mr. Nithyanandam C P and Srinivasan A for assistance in obtaining prototype and test data.

REFERENCES

- [1] Stone, R., 1999, Introduction to Internal Combustion Engines, MacMillan, Chap. 9.
- [2] Watson, N., and Janota, M. S., 1982, Turbocharging the internal combustion engine, MacMillan, Chap. 2.
- [3] Japikse, D., and Baines, N. C., 1997, Introduction to Turbomachinery, Concepts NREC, Wilder, VT, Chap. 4.
- [4] Japikse, D., 1996, Centrifugal Compressor design and performance, Concepts NREC, Wilder, VT, Chap. 6.
- [5] Lewis, R., I., 1996, Turbomachinery Performance Analysis, Arnold Press, Chap. 7.
- [6] Swain, D., 2014, "Impact of Impeller Blade Trimming on the Performance of Centrifugal Compressors," Ph.D. thesis submitted to Dept. Mech. Eng., Michigan State University.
- [7] Wang, T., Peng, C., Wu, J., 2014, "Back Swept Angle Performance Analysis of Centrifugal Compressor," *Mechanika*, 20 (4), pp. 402–406.
- [8] Xu, C., and Amano, R. S., 2012, "Meridional Considerations of the Centrifugal Compressor Development," *Int. J. Rotating Machinery*, Vol. 2012, Article ID. 518381.
- [9] Tamaki, H., Unno, M., Kawakubo, T., and Hirata, Y., "Aerodynamic Design to Increase Pressure Ratio of Centrifugal Compressors for Turbochargers," ASME Paper, GT2009-59160.
- [10] Chen, T., Zhang, Y., Zheng, X., and Zhuge, W., 2010, "Compressor design for multi-point operating conditions of turbocharged gasoline Engines," ASME Paper No. GT2010-22283.
- [11] Demeulenaere, A., Bonaccorsi, J., Gutzwiller, D., Hu, L., and Sun, H., 2015, "Multi-disciplinary Multi-point optimization of a Turbocharger Compressor Wheel," ASME Paper No. GT2015-43631.
- [12] Perrone A., Ratto L., Ricci G., Satta F., and Zunino P., 2016, "Multi-disciplinary Optimization of a Centrifugal Compressor for Micro Turbine Applications," ASME paper No. GT2016-57278.
- [13] Diener, O. H. F., Van der Spuy, S. J., Von Backstrom, T. W., and Hildebrandt, T., 2016, "Multi-disciplinary Optimization of a Mixed-flow Compressor Impeller," ASME paper No. GT2016-57008.
- [14] AxSTREAM, SoftInWay Inc.
- [15] ANSYS-CFX & Mechanical V16.2, Ansys Inc.
- [16] Aungier, R., H., 2000, Centrifugal Compressors: A Strategy for Aerodynamic Design and Analysis, ASME Press.
- [17] Spalart, P. R., and Shur, M., 1997, "On the Sensitization of Turbulence Models to Rotation and Curvature," *Aerospace Science and Technology*, 1 (5), pp. 297–302.
- [18] Ross, P., J., 2005, Taguchi Techniques for Quality Engineering, Tata McGraw-Hill Publishing Company Ltd.
- [19] Moroz, L., Govoruschenko, Y., Pagur, P., and Romanenko, L., 2008, "Integrated Conceptual Design Environment for Centrifugal Compressors Flow Path Design," IMECE2008-69122.
- [20] BorgWarner Inc.
- [21] Bohn, D., Heuer, T., and Kusterer, K., 2005, "Conjugate Flow and Heat Transfer Investigation of a Turbo Charger," *ASME J. Eng. Gas Turbines Power*, 127(3), pp. 663-669.
- [22] Baines, N., Wygant, K. D., and Dris, A., 2010, "The Analysis of Heat Transfer in Automotive Turbochargers," *ASME J. Eng. Gas Turbines Power*.
- [23] Argali, H., and Angstrom, H., 2012, Turbocharger SI Engine Simulation with Cold and Hot-Measured Turbocharger Performance Maps, ASME Paper GT2012-68758.
- [24] MINITAB Statistical Software V17.