CFD Analysis of an Automotive Turbocharger for Enhanced Engine Performance

Ashish Singh, Archit Sasane, Rohan Patney, and R. Harish

Abstract In recent years, developments of automobile downsizing promote the developers to enhance the performance of current turbocharging technology. Turbocharger has become one of the key components in the automotive industry as it helps to enhance the engine performance. Due to drawbacks of conventional radial turbine used in turbocharging techniques, preliminary design of axial turbine was proposed, in order to achieve highest performance of turbocharger axial turbine and therefore enhance the engine performance. In the present study, the optimal design is made based on the NACA profile blade of a single axial turbine for the turbocharger system on solidworks. A computational fluid dynamics (CFD) analysis is carried out, and the turbine design is modified based on the analysis results in order to attain optimum performance and minimal lag. The CFD analysis results of the velocity and pressure distributions identified the flow behaviour patterns such as flow separation, vortexes, and performance characteristics.

Keywords Axial turbine · Turbocharger · Computational Fluid Dynamics (CFD) · Volute · Compressor

1 Introduction

Turbocharger is a device which is the integral part of an internal combustion (IC) engine, which is widely used to enhance the engine performance. The compressor is driven by the impeller which is in turn driven by the exhaust gases. Presently, this technology is majorly utilized in automotive and aerospace industries. Factors like improved engine performance and fuel efficiency, government regulations, and engine downsizing have increased the need for turbocharging technologies. Computational fluid dynamics (CFD) may be a cost affected tool to supply detailed flow information inside the entire turbocharger. The research conducted by Chehhat et al. [1], Lintz [2] and Yang et al. [3] showed that volute significantly influences the overall

A. Singh · A. Sasane · R. Patney · R. Harish (✉)
School of Mechanical Engineering, Vellore Institute of Technology, Chennai, Tamil Nadu 600127, India
e-mail: harish.r@vit.ac.in

A. Maurya et al. (eds.), *Recent Trends in Mechanical Engineering*, Lecture Notes in Mechanical Engineering, https://doi.org/10.1007/978-981-19-7709-1_69

performance such as the operating range, the stability, and the location of the best efficiency point of the compressor. Not just the compressor impeller but the volute also affects the overall performance. So, it was necessary to include its effect in our study. Lintz et al. [2] went further and studied the significance of volute surface roughness and found that it had a significant impact on the automotive turbocharger performance. Improving the surface quality of the volute has potential benefits in turbine efficiency. In the research performed by Rinaldi [4] experiments and simulations were carried out to understand the performance of the CO_2 compressor and the CFD and experimental results were compared by addressing both the limitations of the adopted models, and those related to the experimental data. In this study [5], the researchers investigated different CFD modeling settings about interfaces and boundary conditions to successfully model compressible flow in various types of high-speed compressors. In the study carried out by Le. Sausse et al. [6], the compressor maps are obtained experimentally and then compared with CFD results. Several speed lines are calculated by varying input conditions. Galindo et al. [7, 8] carried out numerical investigations to understand the significance of geometry on the automotive turbocharger performance. Zheng et al. [9] studied and investigated the surge and stall conditions with a vaned diffuser by experiments. Jawad et al. [10] in their study proposed a preliminary design of axial turbines due to the drawbacks of conventional radial turbines used in turbochargers. A CFD analysis is carried out and is compared with the conventional radial turbine analysis results. Turbocharger's main function is to increase the efficiency of the engine [11]. In turbocharger the more the amount of air has been hit to an impeller and even by doing modification with the intake geometry the performance is enhanced significantly [12]. In the research paper done by Nicholas Anton et al. on axial turbochargers, they found that these types of turbochargers have less moment of inertia and also the design is wider than the conventional radial turbochargers and due to their more efficiency at higher flow rates which helps to get the most optimized solution [13–17]. In the research conducted on axial turbocharger by Serrano et al. [14] it is found that the tip leakage loss is directly proportional with the blade loading, whereas in case of radial turbocharger it is directly proportional to the rotational speed. Due to increase in chamber roughness the efficiency of the axial turbocharger is reduced nonlinearly [15]. Even though here we have seen the reasons for the use of axial turbochargers but in the study conducted by Jawad et al. [16] it has been found that the multi-stage axial turbochargers are more efficient and they increase the performance of the engine to a whole new level. In the present work, a turbocharger compressor along with the housing used in a DI diesel engine was designed using Solidworks. We have decided to select a backward curved Blade design for our compressor impeller. Backward curved blade has the highest efficiency. The benefit of using a backward curved impeller is that it does not have a stall point on its characteristic. This means that there is no point on the fan characteristic curve that should not be operated. A CFD flow analysis was done on the compressor blades at different blade speeds (rpm) and the results were analysed in order to give out maximum boost thereby enhancing the engine performance.

2 Methodology

In this paper, the work was divided into mainly three parts, the first was the making of a model which was done in the solidworks software. The mesh was generated on volume extraction of volute and Boolean subtraction on the blades collectively on Ansys Workbench. Here, fine mesh sizing was used which generated 1,429,503 cells in total and 420,115 nodes on both volute and blades. This model was imported to Ansys fluent where the simulations were calculated for a pressure-based solver, where the blades were set to a moving wall at the speed of 20,000 rpm and 40,000 rpm respectively in order to compare both simulations simultaneously. We have considered a low rpm range after studying surge and choke limits of various automobile turbochargers. These simulations were performed at three different inlet velocities, i.e. 10 m/s, 25 m/s, and 50 m/s. Figure 1 represents the geometrical configuration of the turbocharger compressor and the generated mesh using Ansys.

2.1 Governing Equations

$$\frac{\partial\rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho u)}{\partial y}+\frac{\partial(\rho u)}{\partial z}=0 \tag{1}$$

$$\begin{aligned}\frac{\partial(\rho u)}{\partial t}+\frac{u\partial(\rho u)}{\partial x}+\frac{v\partial(\rho u)}{\partial y}+\frac{w\partial(\rho u)}{\partial z}&=-\frac{\partial p}{\partial x}\\+\frac{\partial}{\partial x}\left(\frac{\mu\partial u}{\partial x}\right)+\frac{\partial}{\partial y}\left(\frac{\mu\partial u}{\partial y}\right)+\frac{\partial}{\partial z}\left(\frac{\mu\partial u}{\partial z}\right)&-S_x\end{aligned} \tag{2}$$

$$\begin{aligned}&\frac{\partial(\rho u)}{\partial t}+\frac{u\partial(\rho u)}{\partial x}+\frac{v\partial(\rho u)}{\partial y}+\frac{w\partial(\rho u)}{\partial z}=-\frac{\partial p}{\partial y}+\frac{\partial}{\partial x}\left(\frac{\mu\partial v}{\partial x}\right)\\&+\frac{\partial}{\partial y}\left(\frac{\mu\partial v}{\partial y}\right)+\frac{\partial}{\partial z}\left(\frac{\mu\partial v}{\partial z}\right)-S_y\end{aligned} \tag{3}$$

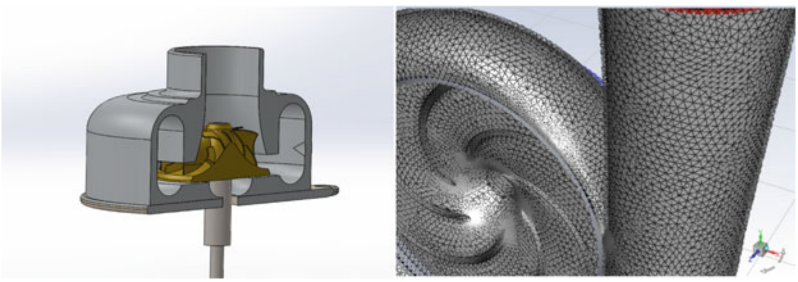

Fig. 1 Turbocharger compressor geometric model and generated mesh

$$\frac{\partial(\rho u)}{\partial t} + \frac{u\partial(\rho u)}{\partial x} + \frac{v\partial(\rho u)}{\partial y} + \frac{w\partial(\rho u)}{\partial z} = -\frac{\partial p}{\partial w} + \frac{\partial}{\partial x}\left(\frac{\mu\partial w}{\partial x}\right) + \frac{\partial}{\partial y}\left(\frac{\mu\partial w}{\partial y}\right) + \frac{\partial}{\partial z}\left(\frac{\mu\partial w}{\partial z}\right) - S_z \tag{4}$$

$$\frac{\partial k}{\partial t} + U_j\frac{\partial k}{\partial t} = P_k - \beta^* k\omega + \frac{\partial k}{\partial t}\left[(v + \sigma_k v_T)\frac{\partial k}{\partial t}\right] \tag{5}$$

Above mentioned are the few governing equations which are solved in each iteration to create the simulation. Equation (1) is the equation to solve continuity. Equations (2), (3) and (4) are the momentum equations on the respective axes, namely x, y and z direction. Finally, the Eq. (5) is the energy equation for the k-ω viscosity model equation.

2.2 Boundary Conditions

The pressure and temperature boundary conditions at the compressor inlet are 10 m/s specified with standard atmospheric conditions. The flow is axial to the impeller at 10 m/s, 25 m/s, and 50 m/s for different cases. At the outlet, static pressure is applied. Subsonic boundary was defined with the SST k-ω turbulence model with 5% turbulence intensity. The mesh motion method is adopted to rotate the impeller blades for a fixed impeller speed. The blades were defined as smooth surfaces moving at a specific speed and the compressor speed was set according to the load on the engine.

3 Results and Discussions

Simulation of the turbocharger was considered for two different impeller speeds (rpm): 20,000 and 40,000 and three different velocities: 10 m/s, 25 m/s, and 50 m/s. Velocity and pressure simulation results were obtained using CFD and the contours and graphs were analysed for both impeller speed and shown in Figs. 2 and 3.

3.1 Velocity Simulation

Following figure shows the velocity vector contour. The velocity is maximum at the compressor wheel blade endings in all the cases. The increased velocity is slowed down in the volute and due to the volute a swirl motion is generated. Figures 4 and 5 shows the air velocity contours at lower inlet speed of 10 m/s. Figures 6 and 7

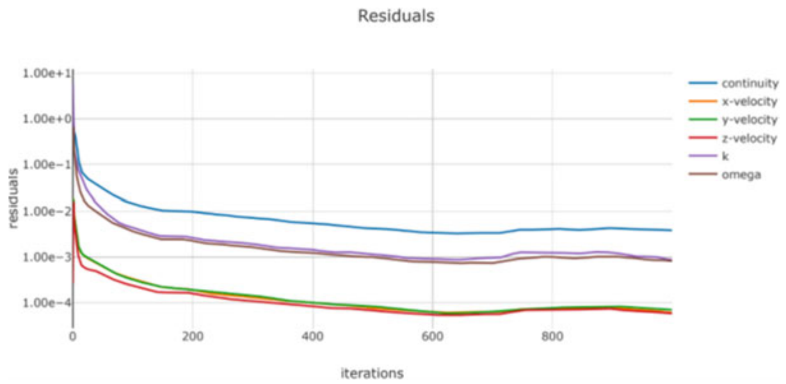

Fig. 2 Variation of residuals for an impeller speed of 20,000 rpm

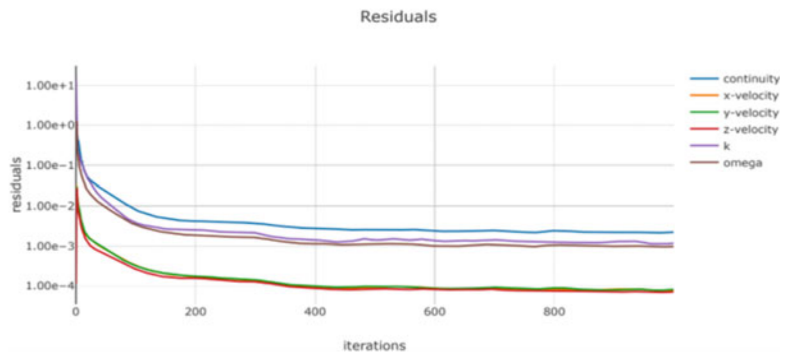

Fig. 3 Variation of residuals for an impeller speed of 40,000 rpm

indicates the air velocity distribution at transitional speeds of 25 m/s and 50 m/s. Figures 8 and 9 shows the velocity contours at a higher rotational speed of 40,000 rpm.

3.2 Pressure Simulation

The contours of the pressure distribution at the assembly of the compressor wheel and housing are investigated for different inlet speeds. The highest pressure is found in the downstream of the compressor wheel and the lowest upstream—suction opening of the compressor wheel. The pressure distribution looks symmetric and is similar for all the cases. Figures 10 and 11 shows the pressure distribution at a lower inlet speed of 10 m/s. Figures 12 and 13 indicates the pressure variations at transitional inlet speed of 25 m/s and 50 m/s respectively. Figures 14 and 15 shows the pressure contours at

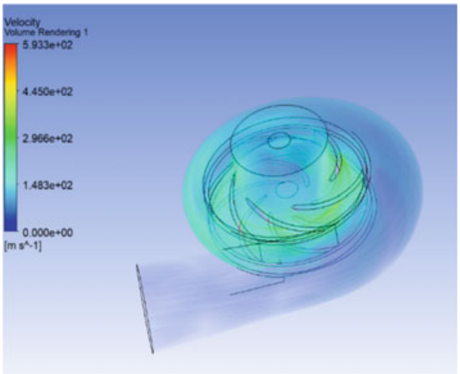

Fig. 4 Air velocity contours at 20,000 rpm, inlet speed 10 m/s

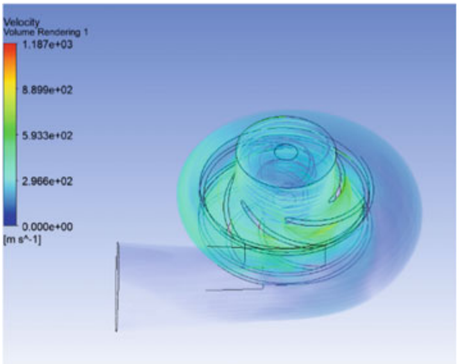

Fig. 5 Air velocity contours at 40,000 rpm, inlet speed 10 m/s

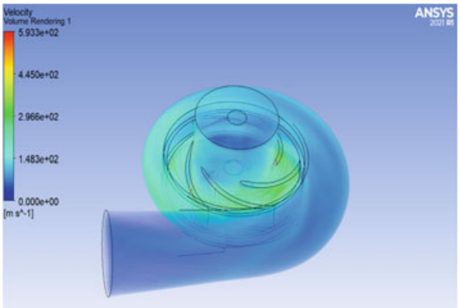

Fig. 6 Air velocity contours at 20,000 rpm, inlet speed 25 m/s

a higher rotational speed of 40,000 rpm. Figure 16 indicates the particle streamlines representing the development of swirl motion inside the turbocharger.

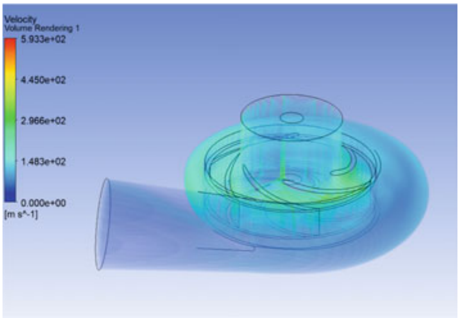

Fig. 7 Air velocity contours at 20,000 rpm, inlet 50 m/s

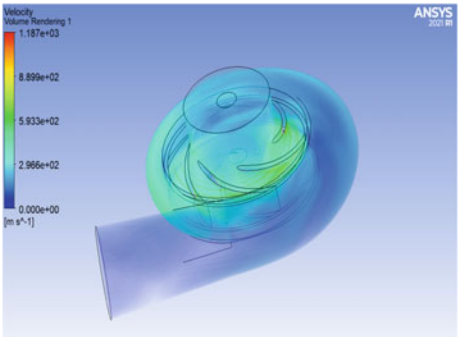

Fig. 8 Air velocity contours at 40,000 rpm, inlet speed 25 m/s

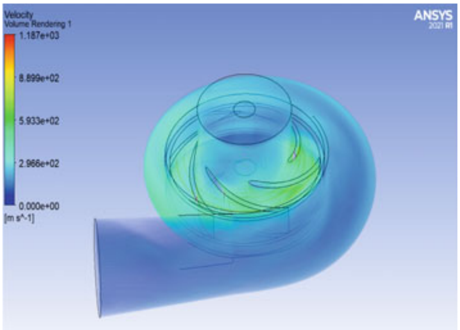

Fig. 9 Air velocity contours at 40,000 rpm, inlet 50 m/s

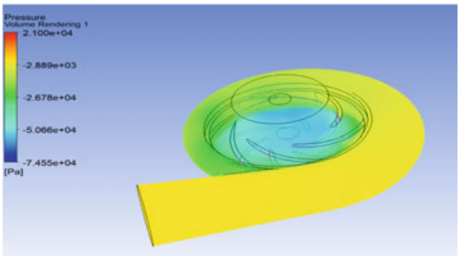

Fig. 10 Pressure contours at 20,000 rpm, inlet speed 10 m/s

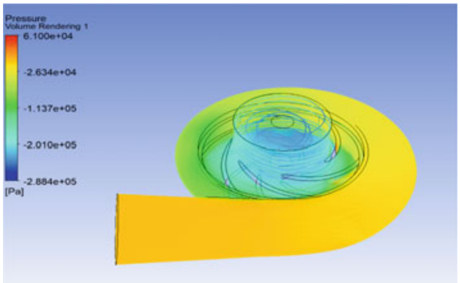

Fig. 11 Pressure contours at 40,000 rpm, inlet speed 10 m/s

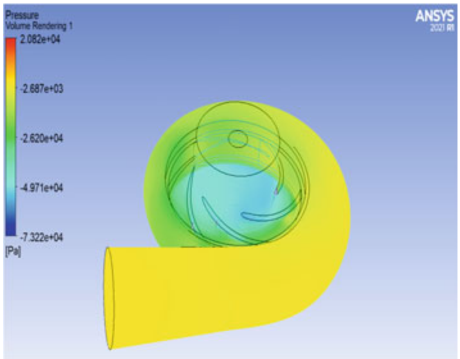

Fig. 12 Pressure contours at 20,000 rpm, inlet speed 25 m/s

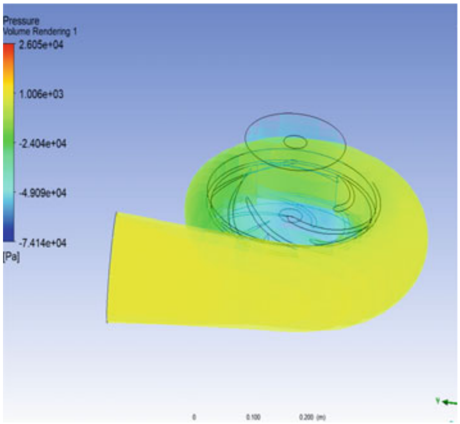

Fig. 13 Pressure contours at 20,000 rpm, inlet speed 50 m/s

4 Conclusions

This paper presents an effort to model and analyse the flow from inlet to the exit of an automobile turbocharger compressor with all the components in place. The flow was analysed at two different compressor blade speeds, i.e. 20,000 and 40,000 and three

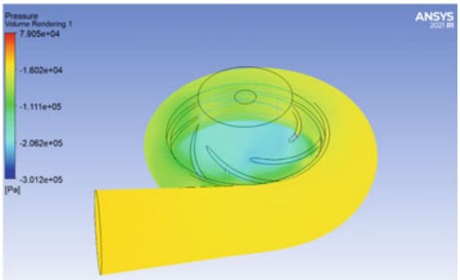

Fig. 14 Pressure contours at 40,000 rpm, inlet speed 25 m/s

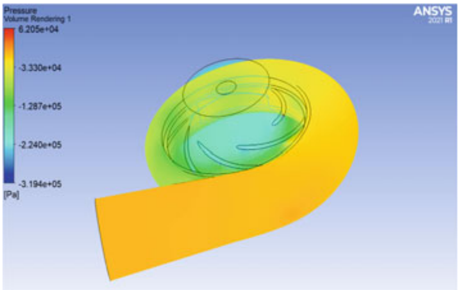

Fig. 15 Pressure contours at 40,000 rpm, inlet speed 50 m/s

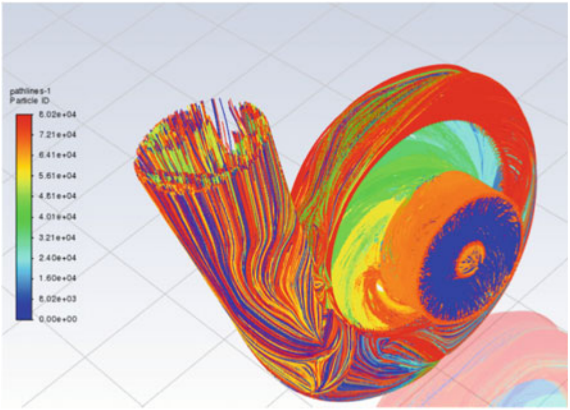

Fig. 16 Particle streamline showing swirl motion at volute exit

different inlet velocities, i.e. 10 m/s, 25 m/s, and 50 m/s. Using the computational fluid dynamics (CFD) gave a better understanding of the behaviour of flow through a turbocharger compressor and how it impacts the turbocharger efficiency and how the turbocharger compressor performs in an automobile engine for enhancing its performance. The velocity simulation revealed that the velocity is maximum at the compressor blade endings and it is slowed down in the compressor volute and swirl motion is generated due to the volute. A pressure increase is seen at the compressor exit compared to the inlet. The highest pressure is found downstream the compressor wheel volute discharge. High pressure at the exit results in extra compressed air for the IC engine thereby increasing its power output.

References

1. Abdelmadjida C, Mohamedb S-A, Boussadc B (2013) CFD analysis of the volute geometry effect on the turbulent air flow through the turbocharger compressor, terragreen 13 international conference 2013
2. Lintz A (2017) The effect of volute surface roughness on the performance of automotive turbocharger turbines. In: Global power and propulsion forum, GPPF 2017
3. Yang M, Martinez-Botas R, Rajoo S, Yokoyama T, Ibaraki S (2015) An investigation of volute cross-sectional shape on turbocharger turbine under pulsating conditions in internal combustion engine. Energy Convers Manag
4. Rinaldi E, Pecnik R, Colonna P (2014) Computational fluid dynamics simulation of a supercritical CO2 compressor performance map. J Eng Gas Turbines Power
5. Schreiber J, Ottavy X, Ngo Boum G, Aubert S, Sicot F (2015) Numerical simulation of the flow field in a high speed multistage compressor- study of the time discretization sensitivity. In: Turbine technical conference and exposition GT
6. Le Sausse P, Fabrie P, Arnou D, Clunet F (2013) CFD comparison with centrifugal compressor measurements on a wide operating range. EDP Sci
7. Galindo J, Gil A, Navarro R, Tari D (2019), Analysis of the impact of the geometry on the performance of an automotive centrifugal compressor using CFD simulations. Appl Thermal Eng
8. Galindo J, Tiseira A, Navarro R, Tari D, Meano CM (2017) Effect of the inlet geometry on the performance, surge margin and noise emission of an automotive turbocharger compressor. Appl Thermal Eng
9. Zheng X, Sun Z, Kawakubo T, Tamaki H (2017) Experimental investigation of surge and stall in a turbocharger centrifugal compressor with vaned diffuser. Exp Therm Fluid Sci
10. Jawad LH, Razzaq HY, Hasan HM (2020), Aerodynamic design and performance investigation of an axial turbocharger turbine for automotive applications. Period Eng Nat Sci
11. Panayides C, Pesyridis A, Saravi SS (2019) Design of a sequential axial turbocharger for automotive application. Energies
12. Ramachandran D, Somashekarappa S, Mayandi B, Reddy Shanmugam R, Boolingam S, Ashok G (2018) A study on the influence of intake geometry on the turbocharger compressor performance, ASME
13. Anton N, Genrup M, Fredriksson C, Larsson PI, Christiansen-Erlandsson A (2018) Axial turbine design for a twin-turbine heavy-duty turbocharger concept, ASME
14. Serrano JR, Navarro R, García-Cuevas LM, Inhestern LB (2018) Turbocharger turbine rotor tip leakage loss and mass flow model valid up to extreme off-design condition with high blade to jet speed ratio, Elsevier
15. Wang X, Zhang X, Zuo Z, Zhu Y, Li W, Chen H, Ding Y (2021) Effect of chamber roughness and local smoothing on performance of a CAES axial turbine, Elseveir
16. Jawad LH, Hasan HM, Razzaq HY (2020) Performance evaluation of a multi-stage axial flow turbocharger turbine, solid state technology
17. Berchiolli M, Guarda G, Walsh G, Pesyridis A (2019) Turbocharger axial turbines for high transient response. Appl Sci