Numerical Investigation of a Turbocharger Compressor in Different CFD Codes and Validation with Experiments

Mert Alpaya and Iskender Kayabasi
Borusan Technology Development and R&D Inc., Istanbul, 34750 Turkey
Email: malpaya@borusan.com; ikayabasi@borusan.com

Burag Hamparyan
Supsan Automative Components Inc., Istanbul, 34295 Turkey
Email: bhamparyan@supsan.com

***Abstract*—Compressor's pressure ratio (PR) is the main parameter that indicates turbocharger's performance. Some tests are performed in order to measure pressure ratio and map the compressor's performance. However, limits of test setups might not be appropriate for testing all performance scenarios in different operating conditions. Additionally, prototype production costs and tests on all conditions can be very expensive and time consuming. Hence, it is more economical and appropriate to measure the performance of turbocharger compressor via numerical methods. The purpose of this paper is to create numerical models of centrifugal turbocharger compressor in different Computational Fluid Dynamics (CFD) codes and validate the models with experimental data.**

***Index Terms*—centrifugal compressor, computational fluid dynamics, moving reference frame, turbocharger**

I. Introduction

Turbochargers have importance in today's automotive industry as they make it possible to obtain more power output from an internal combustion engine while decreasing fuel consumption and CO_2 emission. Turbochargers consist of two main components: compressor and turbine. When engine starts to work, exhaust gas drives turbine which is located at the rear side of turbocharger. Compressor is also driven via common shaft between turbine and compressor. As compressor wheel rotates, it takes and compresses intake air before it enters the combustion chamber. By the virtue of compression, density of the air increases, so that more air is sent to combustion chamber for combustion process and more power is generated. Due to severe restrictions on efficiency and emission rate, turbochargers have been widely used in automotive industry.

The main parameter that indicates turbocharger performance is compressor's pressure ratio. In order to measure pressure ratio and map the compressor's performance, some tests are performed. However, limits of the test setup might not be appropriate for testing all performance scenarios in different operating conditions of the turbocharger. Therefore, it is more economical and appropriate to measure the performance of turbocharger compressor via numerical methods.

Turbocharger compressor performance has been the subject of a large number of numerical and experimental studies. In the study of Baris and Mendonça [1], turbocharger compressor performance characteristics between 100,000 and 200,000 rpm (revolutions per minute) values were investigated numerically in steady state and compared with rig measurements. Polyhedral volume mesh was used with tetrahedral boundary layer mesh in the entire flow model and turbulence was modelled with k (turbulent kinetic energy)-ω (specific rate of dissipation)-SST (Shear Stress Transport) model. Total pressure-total temperature was applied as inlet boundary condition and static pressure was applied as outlet condition. Sufficient accuracy level was achieved within 2% difference of rig measurements. Bhardwaj and Gupta [2] analyzed a centrifugal compressor of a diesel engine at different design speeds in their study. Compressor simulation was composed of two steps: impeller and diffuser. A stage type mixture interface was used in order to connect impeller outlet and diffuser inlet. Total pressure and total temperature values were applied as inlet boundary conditions. Numerical results were compared and validated with available test data. Prasad *et al*. [3] numerically analyzed a centrifugal compressor for different mass flow rate and rotational speed values. CFD results were compared and validated with experimental data for chosen performance parameters like stage input power, polytrophic efficiency and pressure ratio. Stage type mixing interface was used to connect impeller outlet and diffuser inlet. Total pressure boundary condition was used at the inlet of the impeller. Abdelmadjid *et al*. [4] showed that volute geometry has considerable impact on the pressure and temperature values at the compressor outlet. Three different volute designs with same impeller and diffuser were numerically analyzed at 100,000 rpm with steady conditions and compared with each other. Effects of the shape of volute cross section and the

Manuscript received March 3, 2018; revised June 25, 2018.

doi: 10.18178/ijeetc.7.3.135-140

location of volute inlet on overall performance and operating range were investigated. In the study of Le Sausse *et al*. [5], centrifugal compressor performance test results were compared with CFD results on a wide operating range. Total pressure and total temperature values were used as inlet boundary conditions with static pressure outlet condition. In order to obtain mesh refinement, 8 different meshes with different qualities were prepared and solved. CFD simulations made it possible to understand what is happening while approaching surge without testing. Çanga *et al*. [6] compared CFD simulations and test results of a turbocharger compressor in their study. Compressor performance map was created by tests performed between 60,000 and 150,000 rpm rotational speed values. On the other hand, CFD studies were carried out for four different operating conditions at 120,000 rpm. Numerical results showed better similarity with test results at low flow rate values while deviation was increased at higher flow rates. Li *et al*. [7] investigated centrifugal performance of turbocharger numerically in their study. In addition, effect of mesh number and distribution on numerical results of compressor also investigated and validated with test data. Numerical results showed similarities with experimental results which indicate CFD method's convenience. In the numerical study of Jawad *et al*. [8], the effect of double splitters on a modified turbocharger compressor performance was investigated. Polyhedral mesh structure was used for volume mesh generation and turbulence was modelled with k-ω-SST model. Total pressure and total temperature boundary condition was set at the inlet and static pressure was set at the outlet. The parametric simulations showed that the potential of double splitter in improving centrifugal compressor performance. Liu [9] investigated the effect of impeller inlet angle of the compressor and average impeller thickness on compressor efficiency. Simulations were carried out at steady state, adiabatic and compressible turbulent flow. Spalart-Allmaras equation model was used for turbulence modeling. Atmospheric pressure inlet and mass flow outlet boundary conditions were applied to model at 180,000 rpm. Optimum impeller inlet angle and average impeller thickness values were obtained with simulations. Validation of CFD model was done with experimental results. In the study of Moussavi *et al*. [10], the effect of position and angle of splitters on compressor's performance was investigated. Simulations were performed with total pressure inlet and mass flow rate outlet boundary conditions. Frozen rotor method was used to model rotational movement of the impeller and SST model was implemented in order to model turbulence in the flow. Li and Spence [11] investigated the effect of the axial position of multiple reference frame stator-rotor interface between the inlet guide vanes and impeller on predicted flow field. Total pressure inlet and static pressure outlet boundary conditions were applied to prepared numerical model and the turbulence was modeled with SST turbulence model. Dickman *et al*. [12] investigated a turbocharger compressor by means of CFD in transient conditions in order to analyze vibration of impeller blades. They applied k-ε turbulence model with scalable wall functions as turbulence model and they used total pressure and total temperature boundary condition at the inlet and static pressure at the outlet. In the study of Ding *et al*. [13], CFD was used to investigate a centrifugal compressor's off-design performance. Total pressure and total temperature inlet boundary condition was used with mass flow rate outlet condition. Turbulence modelling was done with SST turbulence model. Shahin *et al*. [14] investigated the performance of a centrifugal compressor that operating near surge condition by means of CFD. They used mass flow rate inlet and pressure outlet boundary conditions with RNG k-ε turbulence model. Simulation that they performed was transient in order to get pressure and velocity data for different time steps.

The purpose of this paper is to create numerical models of the selected turbocharger compressor in two different commercial CFD codes and validate the results with experimental studies at different rotational speeds and pressure ratios. Six different experiment cases, varying from 50,000 rpm to 105,000 rpm and from PR=1.05 to PR=1.345, were set and performed for obtaining boundary conditions and validating numerical results. Numerical studies were conducted in steady-state conditions with mass flow rate inlet and pressure outlet boundary conditions. Besides, rotation of compressor wheel was modeled with rotating (moving) reference frame method while turbulence was being modeled with standard k (turbulent kinetic energy) - ε (turbulent dissipation) and Spalart-Allmaras turbulence models. Models were solved both in Cartesian and polyhedral mesh structures by means of two different codes. Convergence of the solutions were monitored by creating surface goals for density, static pressure, mass and volumetric flow rates at the outlet of compressor. Obtained numerical solutions were compared with each other and validated with experimental results for each experiment cases. By creating the validated numerical models of the turbocharger compressor, limited scale of performance test setups was surpassed. Therefore, it has become possible to measure the performance of turbocharger compressor numerically in operating conditions of vehicle and the necessity for toilsome and expensive performance tests was abolished.

II. TEST SETUP

Turbo test flow machine is a device, which is used to measure, check and adjust various turbochargers' performance. In this study, a turbo test flow machine has been used for testing compressor pressure ratio, which has been calculated with compressor outlet pressure divided by compressor inlet pressure. In addition, turbo test machine could measure turbine pressure and flow rate, compressor flow rate and turbocharger rpm speed. General view of the test setup is shown in Fig. 1.

Test device should simulate operating conditions of turbocharger in order to obtain realistic performance results. For this reason, tests were performed according to operating conditions. Besides, oil lubrication was set with suitable temperature and pressure for simulating real time conditions.

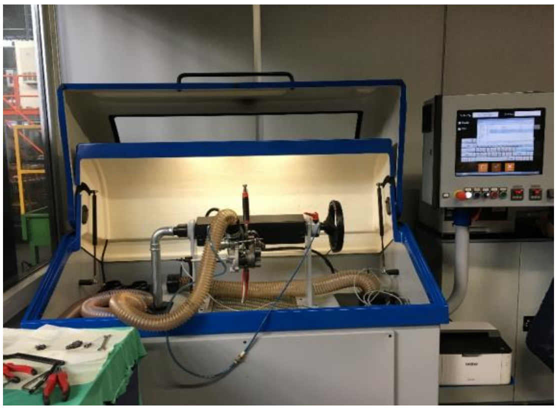

Figure 1. General view of the test setup.

Turbo assembly is important parameter for performance measurement. Turbo should be assembled to adapter plate correctly. Rotating beam holds turbo with adapter. Hydraulic line and oil supply should be inserted to oil inlet for proper lubrication. Pressure ratio and compressor flow rate measurement pipes have been installed to compressor housing outlet. For measuring rpm, rotation measurement sensor has been installed to compressor housing inlet. Rotation measuring device can be seen in Fig. 2. After installation turbocharger and measurement kits, the system becomes ready to start for the test. The pressurized air has been delivered to turbine housing through the steel flow pipe. The air passes the adapter plate port and drives the turbine impeller. Due to rotation, the air leaves turbine housing. Turbine impeller transfers rotational motion to compressor impeller by means of the shaft. Ambient air is sucked through compressor housing due to compressor impeller rotation. Compressor increases the pressure of air by applying work on it. During the test, all measurement values could be observed in real time from control panel and could be saved to database for further researches.

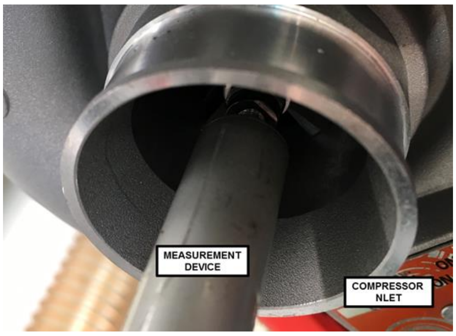

Figure 2. Rotation measurement device.

III. Numerical Modeling

Before running the CFD simulations of turbocharger compressor, 3D geometry data of the compressor and related test equipment were created. Measurement probe of test device is being connected at the outlet of compressor while rotation measurement device is being positioned at the inlet. Hence, in order to correctly model the flow during performance test, inlet and outlet side of compressor geometry was modified. At the inlet, surface area was reduced due to rotation-meter's position.

At the outlet, surface area and outlet length were increased in order to properly measure the outlet properties in fully developed condition. The initial and modified geometry of the turbocharger compressor is shown in Fig. 3.

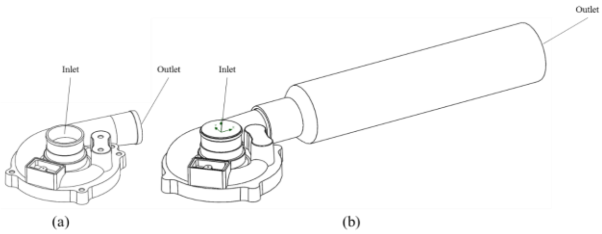

Figure 3. Initial (a) and modified (b) CAD geometry.

Simulations were carried out in FloEFD software at steady state conditions as boundary conditions data for the CFD model were collected when the compressor had reached the steady state. While modeling the turbocharger compressor, mass flow inlet and total pressure outlet boundary conditions with the rotational speed were used. Rotation of the compressor wheel was modeled with rotating reference frame method instead of sliding mesh while turbulence was being modeled with k-ε turbulence model. Test equipment allows to measure volumetric flow rate at the outlet. Therefore, numerical value of volumetric flow rate was calculated in software and compared with test results. Density, static pressure, mass and volumetric flow rate surface goals for outlet were set in order to control the solution's convergence. Calculation was finished when all these surface goals had converged. In FloEFD software, flow region is being defined automatically when the user import CAD model to the software. A fictional rotating body should also be imported to the region where the impeller exists, in order to define rotation to the blades.

In this study, six different test cases of compressor were analyzed. The values for mass flow rate, pressure ratio and rpm measured at the performance tests of the compressor can be seen in Table I for each case. As expected, it was seen that pressure ratio and mass flow rate of the compressor increases with the increasing rotational speed.

TABLE I: Test Cases of Turbocharger Compressor

Rotation [rpm]	PR	Mass Flow Rate [kg/s]
51,670	1.061	0.03222
64,753	1.119	0.04270
86,010	1.199	0.06236
88,217	1.249	0.06704
101,050	1.291	0.08087
104,953	1.342	0.08778

Three different Cartesian type grids were created in order to observe the effect of mesh quality on accuracy and provide mesh independency. Numbers of cells in solution domains of these three grids are 3,088,840, 2,274,290 and 1,462,411 respectively. A sample Cartesian grid for the compressor is shown in Fig. 4.

IV. Grid Independency

Numerical results of turbocharger compressor for three different Cartesian grid structures were obtained while satisfying all specified surface goals at steady state in order to provide grid independency. The results for six different cases for three different mesh structures and their comparison with tests are shown in Table II.

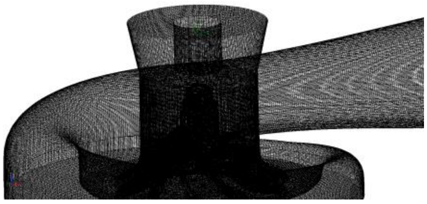

Figure 4. Cartesian grid of the compressor.

TABLE II: Results for Different Cartesian Meshes

Case		Experimental	Mesh-1	Mesh-2	Mesh-3
1	51,670 rpm PR=1.061	0.02520 [m^3/s]	0.02550 (1.19%)	0.02550 (1.19%)	0.02560 (1.59%)
2	64,753 rpm PR=1.119	0.03167 [m^3/s]	0.03240 (2.31%)	0.03250 (2.62%)	0.03250 (2.62%)
3	86,010 rpm PR=1.199	0.04316 [m^3/s]	0.04520 (4.73%)	0.04530 (4.96%)	0.04530 (4.96%)
4	88,217 rpm PR=1.249	0.04454 [m^3/s]	0.04680 (5.07%)	0.04680 (5.07%)	0.04690 (5.30%)
5	101,050 rpm PR=1.291	0.05199 [m^3/s]	0.05550 (6.75%)	0.05580 (7.33%)	0.05570 (7.14%)
6	104,953 rpm PR=1.342	0.05428 [m^3/s]	0.05830 (7.41%)	0.05860 (7.96%)	0.05860 (7.96%)

It can be clearly seen from the Table II that, meshes having different qualities and cell numbers give almost same results at every measurement points. From this point of view, it was proven that simulation results are independent of grid. Compressor's pressure ratio and volumetric flow rate increases with the rotational speed both in tests and numerical models as expected. However, deviation from test results is also being increased with rotational speed. It was considered that numerical and/or testing errors are being increased at high rpms.

V. Tip Clearance Sensitivity

After grid independency, in order to validate the CAD model and check the effect of tip clearance on simulation results, tip clearance sensitivity analysis was performed. The CAD model was prepared by technical draftsmen with the measurements taken from the physical model. Hence, there might be some incompatibility on tolerances of numerical model and physical model. The tolerances of impeller blades and volute determine the clearance of the compressor. In accordance with this purpose, two additional different z-position configurations of compressor wheel CAD model were created. The position of the compressor wheel in volute was first displaced to -0.2 mm in z-direction and then it was displaced to +0.2 mm. Hence, the clearance between compressor blade tip and volute was first increased and analyzed and then it was increased and analyzed. Three different configurations of compressor wheel are shown in Fig. 5.

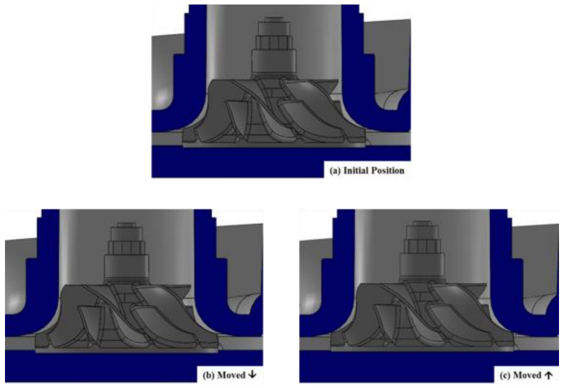

Figure 5. Compressor tip clearance configurations.

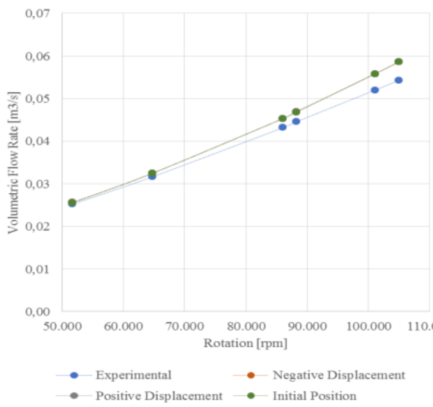

Figure 6. Cartesian grid of the compressor.

As shown in Fig. 5, the clearance between blade tips and volute remained same with initial case (a), increased as the impeller was moved downside (b) and decreased as the impeller was moved upside (c). Two new configurations were analyzed with same boundary conditions of test cases. Once again, numerical value of volumetric flow rate was calculated in software and compared with test results at six different rpm values. Results and comparison of three different clearance configurations (initial position, negative displacement and positive displacement) can be seen in Fig. 6 as a graph for different rotational speed values.

It can be seen from the Fig. 6 that results for each three clearance configurations are very close to each other. Thereby, numerical model was considered as clearance sensitive and results are independent of tip clearance of compressor wheel.

VI. Comparison of CFD Codes

After setting and validating the model in FloEFD commercial code, the model was also set in ANSYS

Fluent in order to see the effects of different mesh types and turbulence models. FloEFD uses standard k-ε turbulence model while modeling the flow by default. Conversely, almost all well-known turbulence models in literature are available in ANSYS Fluent. However, modeling in Fluent needs more care as the flow region is being prepared by the user and meshing is completely user dependent.

Spalart-Allmaras turbulence model was selected among all turbulence models available in Fluent to model the turbulence phenomena in the centrifugal compressor. Spalart-Allmaras is a one-equation Reynolds Averaged Navier-Stokes (RANS) model that was developed for unstructured codes in aerospace industry and it is becoming increasingly popular for turbomachinery applications [15]. One-equation RANS models are based on turbulent kinetic energy (k) transport equation in order to calculate velocity scale ($v=k^{1/2}$).

While modeling the flow in ANSYS Fluent, another type of mesh structure was also used instead of Cartesian mesh. The type of this mesh structure is called “polyhedral”. The polyhedral mesh structure that prepared in ANSYS Fluent for the compressor has 1.736.411 elements and it can be seen in Fig. 7.

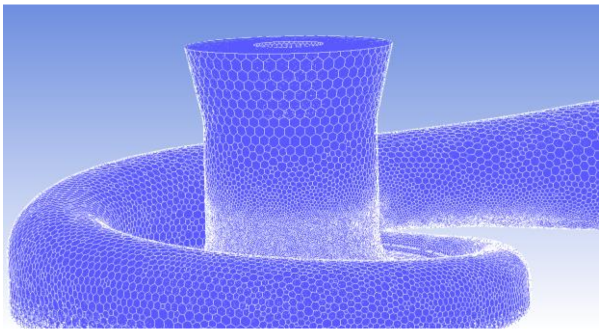

Figure 7. Polyhedral grid of the compressor.

TABLE III: COMPARISON OF DIFFERENT CFD CODES

	Case	Experimental	Code-1	Code-2
1	51,670 rpm PR=1.061	0.02520 [m^3/s]	0.02550 (1.19%)	0.02551 (1.23%)
2	86,010 rpm PR=1.199	0.04316 [m^3/s]	0.04520 (4.73%)	0.04518 (4.68%)
3	104,953 rpm PR=1.342	0.05428 [m^3/s]	0.05830 (7.41%)	0.05832 (7.44%)

After preparing the polyhedral mesh, physics of the problem were set in ANSYS Fluent. The same inlet and outlet boundary condition types and solution convergence criteria were used as in FloEFD. Three of six cases simulated in FloEFD were selected and ran. The comparison of results and test is given in Table III.

As it can be seen from the Table III, two different CFD codes with different mesh structures and turbulence models has shown similar results at various rotational speeds. From this point of view, computational model of the compressor was set in a reasonable way with suitable boundary conditions and CAD geometry. It can be said that model was validated comprehensively.

VII. RESULTS AND DISCUSSION

Model was validated as it deviates maximum 8% of test results between 50,000-105,000 rpm range with different quality meshes and different turbulence models at two different commercial CFD codes. Such a deviation can be acceptable when compared with prior studies in literature. This deviation can originate from modeling and/or experiments and can be minimized by altering solver and/or experiment parameters.

Modeling the flow in the compressor was not only used to validate experimental results but also used to calculate forces acting on compressor blades. These forces were used to investigate the loads in housing region of the turbocharger.

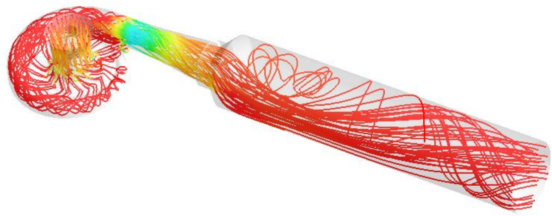

Figure 8. Pressure drop at diffuser of the compressor.

After validating the numerical model, some post-processing studies were carried out in order to properly investigate the turbocharger compressor model. Some design flaws were stand when pressure contours of the model were created. Due to a constriction at the beginning of diffuser section of compressor, there is some pressure drop. This flaw negatively affects the performance of the compressor. The Fig. 8 shows pressure drop at diffuser due to this design flaw.

Therefore, design of the compressor volute needs to be improved in order to increase the performance of turbocharger. Possible design changes on the mentioned turbocharger compressor can be the subject of another paper which contains further steps of this study.

VIII. CONCLUSION

In this study, centrifugal compressor of a turbocharger was modeled and analyzed numerically using two commercial CFD codes. This paper also contains the compressor’s experimental studies which constitute boundary conditions of numerical study and used to validate the models. It can be said that numerical model which consists of mass flow inlet and total pressure outlet boundary conditions under steady conditions with rotating reference frame method and k-ε or Spalart-Allmaras turbulence model is fairly successful to model such a turbocharger compressor. Compressor wheel tip clearance is also an important factor that affects flow properties and should be taken into consideration while modeling such a turbomachinery system.

Next step of this study can be modeling turbine side of the turbocharger also. As hot exhaust gases pass through turbine, it will be more critical to correctly model it. Further step can be model the turbocharger system as whole with hot exhaust gases start to turn shaft and compressor wheel starts to compress inlet air. Such a model requires dynamic mesh modeling with transient calculations.

REFERENCES

[1] O. Baris and F. Mendonça, "Automotive turbocharger compressor CFD and extension towards incorporating installation effects," in *Proc. ASME 2011 Turbo Expo: Turbine Technical Conf. and Exposition*, 2011, pp. 2197-2206.

[2] S. Bhardwaj and K. K. Gupta, "Centrifugal compressor analysis by CFD," *Int. Journal of Science and Research*, vol. 3, no. 11, pp. 475-477, 2014.

[3] V. V. Prasad, M. L. Kumar, and B. M. Reddy, "Centrifugal compressor fluid flow analysis using CFD," *Science Insights: An International Journal*, vol. 1, no. 1, pp. 6-10, 2011.

[4] C. Abdelmadjid, S. A. Mohamed, and B. Boussad, "CFD analysis of the volute geometry effect on the turbulent air flow through the turbocharger compressor," *Energy Procedia*, vol. 36, pp. 746-755, 2013.

[5] P. Le Sausse, P. Fabrie, D. Arnou, and F. Clunet, "CFD comparison with centrifugal compressor measurements on a wide operating range," *The European Physical Journal Conf.*, 2013.

[6] Z. Çanga, E. Abo-Serie, and K. Çarman, "Experimental analysis and CFD simulations of a turbocharger compressor," presented at the 13th Int. Combustion Symp., Bursa, Turkey, 2015.

[7] J. Li, Y. Yin, S. Li, and J. Zhang, "Numerical simulation investigation on centrifugal compressor performance of turbocharger," *Journal of Mechanical Science and Technology*, vol. 27, no. 6, pp. 1597-1601, 2013.

[8] L. H. Jawad, S. Abdullah, R. Zulkifli, and W. M. F. W. Mahmood, "Numerical simulation of flow inside a modified turbocharger centrifugal compressor," *Asian Journal of Applied Sciences*, vol. 5, no. 8, pp. 563-572, 2012.

[9] M. X. Liu, "The CFD analysis of JQ40A gasoline turbocharger compressor," *Applied Mechanics and Materials*, vol. 364, pp. 144-148, 2013.

[10] S. A. Moussavi, A. H. Benisi, and M. Durali, "Effect of splitter leading edge location on performance of an automotive turbocharger compressor," *Energy*, vol. 123, pp. 511-520, 2017.

[11] X. Li and S. Spence, "The impact of the multiple reference frame interface on modelling the interaction between IGVs and the impeller in turbocharger compressors," presented at ASME Turbo Expo 2017: Turbomachinery Technical Conference and Exposition, 2017.

[12] H. P. Dickmann, T. S. Wimmel, J. Szwedowicz, D. Filsinger, and C. H. Roduner, "Unsteady flow in a turbocharger centrifugal compressor: 3D-CFD-simulation and numerical and experimental analysis of impeller blade vibration," in *Proc. ASME Turbo Expo 2005: Power for Land, Sea, and Air*, 2005, pp. 1309-1321.

[13] M. Ding, C. Groth, S. Kacker, and D. Roberts, "CFD analysis of off-design centrifugal compressor operation and performance," dissertation, University of Toronto, 2005.

[14] I. Shahin, M. Gadala, M. Alqaradawi, and O. Badr, "Unsteady CFD simulation for high speed centrifugal compressor operating near surge," in *Proc. ASME Turbo Expo 2014: Turbine Technical Conference and Exposition*, 2014, pp. V02DT44A045-V02DT44A045.

[15] A. Fluent, *13.0 Theory Guide*, *Turbulence*, ANSYS Inc., Canonsburg, PA, 2010.

Mert Alpaya is a Ph.D. student in Mechanical Engineering Department of Istanbul Technical University. He has his MSc degree in Mechanical Engineering (Istanbul Technical University, 2016) and BSc degree in Mechanical Engineering (Istanbul Technical University, 2013).

He has been working as Researcher at Modeling and Simulation Department of Borusan Technology Development and R&D Inc. since September, 2016 in Istanbul. Previously, he was working as R&D Engineer in BSH Home Appliances between 09.2013 – 09.2016. Alpaya's main research interests include computational fluid dynamics (CFD), wind energy and heat transfer.

Iskender Kayabasi has a B.Sc. degree in Aerospace Engineering from Middle East Technical University (METU), Ankara, Turkey, 2009 and MSc degree in aerodynamic from METU at 2012.

He worked as an aerodynamic design engineer at Roketsan, one of the biggest defense industry company in Turkey, from 2009 to 2014. He specialized in aerodynamic design, CFD modeling of dynamic motion, unsteady aerodynamics and aerodynamic heating problems of rockets and missiles. After completing 6 months military service, he worked at GE Aviation as a thermal system design engineer until 2016. He worked for modeling, simulation and design of secondary flows and heat transfer mechanism of aero derivative engines. Since 2016, he has been working as a head of Modeling and Simulation Unit of Borusan R&D Company. Kayabasi's main interest areas are CFD, heat transfer, renewable energy and numerical weather prediction.

Burag Hamparyan has M.Sc. degree in material engineering (Istanbul Technical University, 2016), a BSc degree in metallurgical and material engineering (Yildiz Technical University, 2013). He is PhD candidate at the moment in material science (Yildiz Technical University).

He is R&D Engineer at Supsan Engine Parts since 2016, where he coordinates and conducts R&D projects, manages turbocharger production line, develops new turbocharger references in factory. Hamparyan's main research interests are materials and production techniques of engine valves, design and verification of turbochargers. His personal interest is energy storage devices like super capacitors.