A Comparative Analysis Between Optimized and Baseline High Pressure Compressor Stages

Using Tridimensional Computational Fluid Dynamics

Valeriu Dragan
Computational Fluid Dynamics Department
NRDI Comoti
Bucharest, Romania
valeriu.dragan@comoti.ro

Ion Malael
Computational Fluid Dynamics Department
NRDI Comoti
Bucharest, Romania
ion.malael@comoti.ro

Bogdan Gherman
Computational Fluid Dynamics Department
NRDI Comoti
Bucharest, Romania
bogdan.gherman@comoti.ro

***Abstract*—Re-vamping of industrial turbo-machinery is commonplace in the oil and gas industry in applications where subterranean combustion is used for oil extraction. The current case study refers to such an industrial compressor re-vamping, using a state of the art 3D fully viscous CFD methodology coupled with artificial neural networks (ANNs) and genetic algorithms (GA). The ANN is used to establish correlations within a database of CFD simulations of geometrical variations of the original rotor and the GA uses those correlations to estimate an optimum. The estimate is then tested with the same CFD method and the results are fed back into the database, increasing the accuracy of the ANN correlations. The process is reiterated until the optimum estimated by the GA is confirmed with the CFD simulations. The resulting geometry is superior to the original in terms of efficiency and pressure ratio as well as the range of stabile operation, as confirmed by the successful implementation in the field. In this paper we present an analysis of why the optimized geometry achieves superior performances to the original one. Further work will present comparison between the detailed experimental data and CFD.**

Keywords-optimization; CFD; turbomachinery; centrifugal compressor; artifficial neural network; genetic algorithm

I. INTRODUCTION

Current competiveness requirements in the turbo-machinery industry call for increased operational qualities of the final product. Using modern CFD methods, a tailored made solution, which considers more than one criteria, can be obtained and solution customization can be obtained. This paper presents a re-vamping, optimization case study for a high pressure compressor stage used in the petrol industry. The design theme demanded for a 10 percent increase in mass flow as well as an increase in the isentropic efficiency while maintaining the same outlet diameter, height and rotor aspect ratio. From the start this will imply that the flow coefficient will be increased and that the specific wheel speed is decreased, making it less optimal according to traditional design theories. Since neither the slip factor nor the jet-wake paradigms allow the design of such a stage, unless the models incorporate more elaborate stochastic models, the case study must be carried out using fully viscous numerical simulations. In this particular case, the optimization study was carried out in two steps, firstly using an in-house goal driven method and finally the Numeca Design 3D commercial package.

In the first re-vamping stage, both the rotor and stator were reshaped using conventional 2D methods coupled with the in-house stochastic correlations. This process of the stage re-design also incorporates a primary optimization of the two blade rows. Version 15 (V15) represents the preliminary result of the re -vamping with the Euler in-house code [1]. Intuitively, in designing V15 two key aspects were targeted, namely maintaining the specific speed at the highest possible value while preventing flow separation and, secondly, diminishing the machine Mach number – which was thought as a source of loss. Both those targets were proven to be misguided since V25 actually has a higher machine Mach number and a lower specific speed. Paradoxically it seems that minimizing and maximizing isolated parameters does not lead to the absolute maximum in overall performance.

Although fast in terms of convergence, the in-house code cannot account for various details such as viscous heating of the fluid or boundary layer separation due to curvature effects. Hence the resulting model cannot be considered the peak in terms of performance attainable. In order to quantify these effects, the geometry was tested using the Fine/Turbo numerical solver using a fully structured mesh and the k-ω SST model with curvature compensation. The turbulence model has been chosen over other possible candidates since it is traditionally considered to blend the accuracy of the k- ω model proposed by Wilcox, in the boundary layer, with the stability of the k-ε models in the far field region. Also, the Spalart-Allmaras model was considered in the early stages of the design but it was abandoned in favor of the SST model due to the latter's reliability and validation [2].

A multi-grid method was employed to stabilize the model, increasing the refinement of the mesh as the intermediate solutions were reached. The numerical scheme used is second order upwind and the CFL number of 1. In order to account more accurately for the aerodynamic effects during the compression process, a realistic model of air was used [3].

Because of the possibility that the grid itself can influence the results one way or the other, a preliminary screening was carried out on the V15 design. In this screening a number of meshes, increasing in size were used, while maintaining the same y+ distribution through controlling the first cell size. Results show that for the parameters of interest, i.e. pressure ratio and isentropic efficiency, a mesh of ~2.9 e6 cells produced virtually identical results as the finest mesh tested, but at only a fraction of the computational cost. Table I synthesizes the relevant results for the grid sensitivity tests.

TABLE I. GRID SENSITIVITY TESTS FOR THE RELEVANT PARAMETERS (THE BASELINE VERSION)

Grid	Number of cells	Wall cell	Press. ratio	Isentropic efficiency
Coarse	1341280	1e-3	2.25	82.7 [%]
Intermediate	2143870	1e-3	2.21	82.5 [%]
Fine	2925115	1e-3	2.15	82.2 [%]
Finest	3739729	1e-3	2.15	82.1 [%]

The overall simulation technique is in line with the recommendations in the referenced literature and those of the software producer. Various studies exist in the literature to quantify uncertainty of this methodology [4, 5] but all agree that typically errors lye between 3-5% [6] in most global parameters - which has been confirmed to be the case here as well.

In the final design stage only the rotor geometry was varied, maintaining the same geometry of the diffuser blades. Results of the newly re-vamped compressor stage prove an increase of overall isentropic efficiency from ~80% to 82.2% and, after further optimization using CFD, from 82.2% to 85.35%. For the CFD simulations, the k-omega SST model was used. The final design optimization yielded a slight increase in total pressure ratio and decreasing the outlet total temperature. Improved loading distribution on the rotor is apparent since the axial camber line has reduced back-sweep, increasing the load in that region, while in the radial part, where diffusion typically occurs, the loading is maintained constant. In the two steps of design, the backsweep angle distribution is visible along the entire camberline, particularly in the outermost radial region. After detailed comparative analysis of the optimized geometries it becomes apparent that the camber beta angle influences the entire flow structure, especially the secondary flow types such as the over-the-tip flow and boundary layer flow in the outermost radial region.

An interesting observation regarding the final CFD optimization is that, although the overall efficiency is increased, the actual rotor efficiency decreases. This proves that the overall performance of turbo-machineries is holistic. In other words, there is an optimal way to spread both loading and losses across the stage. An inference is that focusing the optimization on any of the components without observing the overall implications may be misleading. Hence, even if the user assigns degrees of freedom to one component (rotor or stator), the whole stage must be simulated in order to see the global effects. Furthermore, if the stages of a turbo-machine are coupled aerodynamically, they must be included in the simulation as well. The results were applied to the re-vamping of a fully functioning compressor and performed as expected. The industrial compressor re-vamping described in this paper refers only to the fourth, high pressure (HP) stage of machinery used in the oil industry. Since one of the requirements was to keep as much elements as possible from the original machine, the design space was limited to blade setting angles, while maintaining the same rotor height and diameters. Another restriction was the wheel speed which was low, due to technological reasons, even for the original mass flow and pressure ratio. In the first stage of the re-vamping, through flow methods [1, 7] and in-house correlations were used to achieve an improvement of 2.2% in isentropic efficiency and reaching the targeted pressure ratio of 2:1. In order to maintain the robustness of the rotor, only the hub and shroud airfoils were varied and the leading and trailing edges were maintained linear and with no lean. A previous case study [8] has shown that this sort of parametric model allows high isentropic efficiencies to be reached, of above 94% per rotor.

II. THE CFD OPTIMIZATION SETUP

A parametric model first had to be fitted over the pre-designed geometry, establishing the method by which the blades and end-walls are described in three dimensional space by the geometry generator. Based on this, a combined set of 200 individual, slightly varied geometries, was generated with the Latin hypercube sampling method [9, 10], and meshed using a user-defined blocking structure. After insuring that the mesh quality is satisfactory for the individuals, the full viscous Computational Fluid Dynamic simulations were carried out using the baseline results as initial conditions. Variations on the rotor geometry were considered in this case. The relevant flow characteristics were then assessed and interpreted by an artificial neural network which approximated correlations between them and the geometrical characteristics of the individuals. Further, based on these assumed correlations, an optimal geometry, which is in fact a minimization of the imposed penalty function, is predicted and CFD tested using the same initial conditions as in the case of each individual in the database. Since the correlations are typically less than perfect, the per-se optimization is an iterative process, requiring that the CFD data acquired in each optimization step to be added to the database in order to be fed back into the correlation approximation loop. In this case after 20 optimal design loop iterations, the correlation accuracy, compared to the CFD results, became sufficient for the process to be considered converged, as seen in Figure 1.

As presented in [11] the ANN is used to approximate the various influences of the geometrical parameters on the penalty function which needs to be minimized. The ANN is comprised of elementary processing units, or nodes, which are arranged into layers and connected with different weighs in order to create a parallel topology. Two layers were used, one layer for input and one for output layer. The elements of the input vectors are connected to the nodes of the input layer through a weight matrix. Nodes sum the weighted inputs and bias to generate individual scalar outputs which are then processed through a sigmoidal transfer function to the output layer. The

ANN is "trained", using the databases available from the previous CFD simulations, by finding connection weight matrices and bias to make up the output vector. Although typically there needs to be a separate database of CFD runs, which would be used to test the accuracy of the ANN, in this case we generate this database in the form of design loop iterations which are iteratively added to the existing database and the ANN is re-trained.

Numeca's Design 3D uses the learning by back-propagation of the errors algorithm to train the neural network which initializes the training process with small random values for the weight and bias factors. After this, the input vector is fed into the network input and the signal is propagated to the output layer. The resulting output vector is then compared to the desired output vector and the error between them is back-propagated until the error is minimized through adjustments made in the connection weights. These weights are factored by a learning rate coefficient, which in our case has the value of 0.5. The momentum coefficient, with the value 0.8, is set up in order to speed up the convergence of the training process. Also a decay factor may be used in order to avoid exaggerated weight values whish could saturate the sigmoid function, however in this particular case the decay factor was set to 0. Other implementations of the same methodology are presented in [12, 13].

Since the design requirements held more than one criterion, the process was continued for 10 more individuals. This was done in order to compare the “optimal" candidates which, having the same penalty function score, are most convenient for the user's needs. It needs to be said that this last step is not a necessity, since weights can be added to any of the criteria used for the penalty function. However in practice, it is difficult for the weights to be set a priori so that they completely cover all cases - especially for new compressors, since the user doesn't have accurate correlations between the geometry and the flow parameters. Figure 2 shows the meshed geometry of a flow channel for the redesigned baseline.

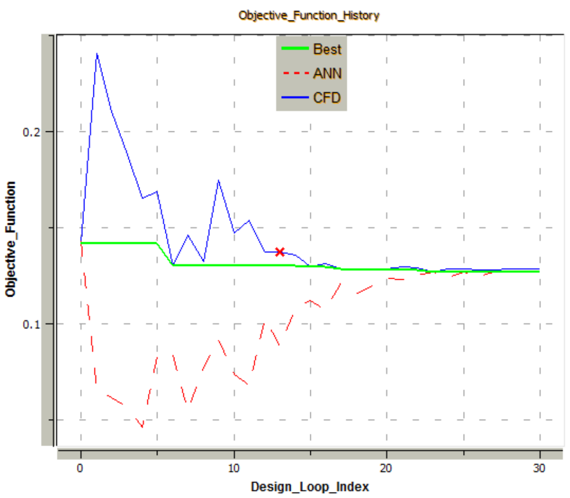

Fig. 1. The ANN approximation of the objective function vs its actual value computed using 3D CFD

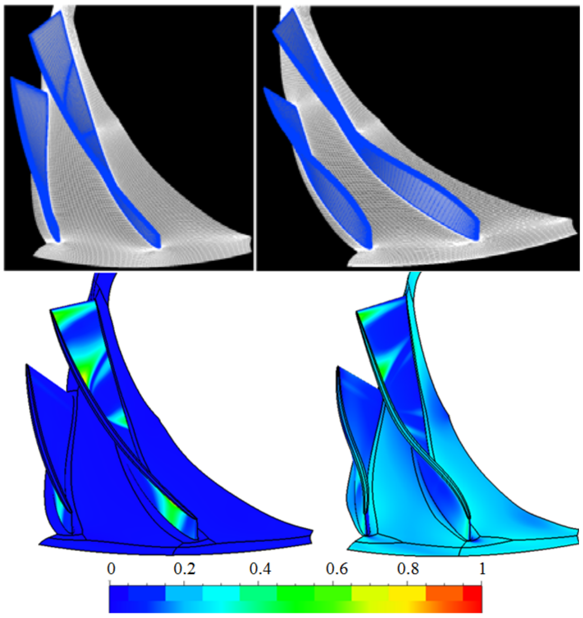

Fig. 2. The meshed geometry of a flow channel for the re-designed baseline and optimized rotor (top) and y+ distribution (bottom)

III. RESULTS AND DISCUSSIONS

In order to make sense of the resulting optimized geometry, both global parameters and flow field analysis were considered. Table II presents a few of the typical turbo machinery parameters calculated for all three cases after the rotor and after the stator. Efficiencies are presented in histogram form in order to provide an easy graphical comparison between cases. One quick observation is that the optimized rotor, V25, has the highest machine Mach number, lowest specific speed and lowest flow coefficient. This may appear as a paradox since most of global parameters correlations [14-19] with the polytropic efficiency suggest the opposite, i.e. higher specific speeds and flow coefficients. However, the machine at hand has a pre-determined rotor aspect ratio and outlet, hence it cannot be fairly compared with designers specifications and/or rules of thumb.

The Cordier diagram [20] suggests that for the aspect ratio of this particular rotor, the specific speed at which it should be operated is low, which is confirmed by the optimization design. It needs to be said that, a further optimization can be achieved by completely re-designing the stage in keeping with the general correlations of [14-19], however this paper only refers to the re-vamping results within the design space. Also from Table II, as well as the histogram below, it can be seen that the pressure ratio of V25 is higher than the other designs, providing a better safety margin in order to compensate the intercooler and pipe total pressure losses.

From Table III, two aspects can be inferred namely that the absolute flow angle, which gives the stator vane incidence, is significantly changed. A second relevant aspect is that the slip factor paradigm only applies, in an approximate manner, to the design point of the rotor. Note that the baseline rotor is functioning at off design point, hence having lower efficiency than its design point. Although the isentropic efficiency is lower on the optimized rotor, the stage efficiency is significantly higher, in spite of the fact that the stator remains unchanged. A simple observation is that the optimized rotor is better suited to the stator. However a more detailed analysis reveals that the polytropic efficiency of V25 is unequivocally higher than that of V15. Since polytropic efficiency only accounts for the aerodynamic losses - not the thermodynamic losses - one might conclude that there is an apparent "barrier" due to the thermodynamic losses. This is because the task the stage must perform is not tailored to the original design - i.e. wheel diameters and speed. Figures 3 to 6 demonstrate different aspects of the design.

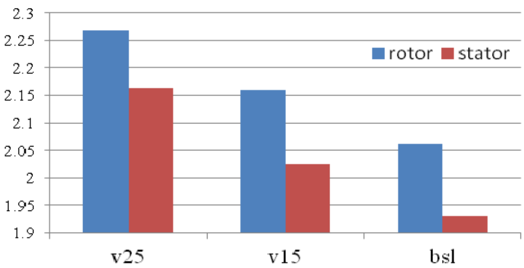

Fig. 3. The total to total pressure ratios of the three designs

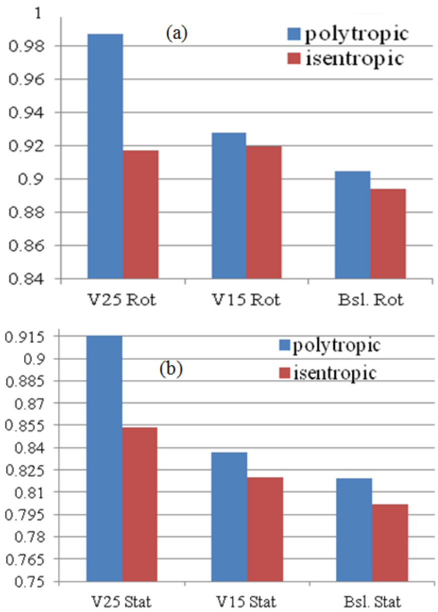

Fig. 4. (a) Total to total isentropic and polytropic efficiency of the rotors and (b) Total to total isentropic and polytropic efficiency of the stages

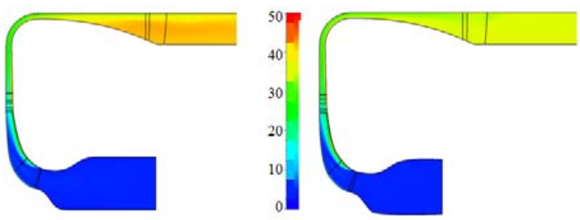

Fig. 5. Entropy [J/(kg*K)] field distribution for the V15 design (left) and V25 (right)

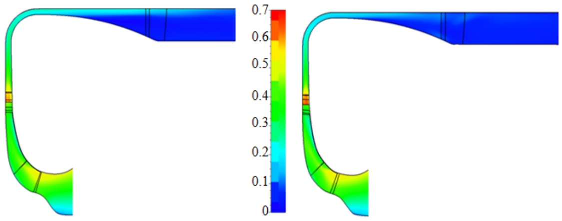

Fig. 6. Relative Mach number field for the V15 design (left) and V25 (right)

TABLE II. GLOBAL PARAMETERS OF THE REVAMPING DESIGNS VS THE BASELINE OFF DESIGN POINT (ROTOR ONLY)

Name	Machine Mach No.	Specific Speed	C1/ U2	W_in/ W_out	W_out [m/s]
Bsl.	0.67	0.51	0.32	1.62	226.60
V25	0.695	0.437	0.282	1.12	160.57
V15	0.659	0.479	0.285	1.22	176.61

TABLE III. MASS AVERAGED FLOW PARAMETERS (ROTOR ONLY)

Name	$V_{throughflow}$ [m/s]	$V_{tangential}$ [m/s]	Absolute flow angle [°]	Slip CFD	Slip Wiesner
V25	73.2	235.2	72.7	0.852	0.878
V15	78.2	220.1	70.5	0.813	0.868
Bsl.	137.4	202.9	59.15	0.62	0.86

An entropy distribution study shows the differences between the two designs, V15 and V25, particularly in the stator region. Figure 7 reveals that the rotor of V25 produces more entropy than V15, however the stator performs significantly better. This can be explained by looking at Figure 6 where the relative Mach number is depicted. Since the stator is identical in both cases, the improvement must be attributed to the better flow angle matching, the higher throughflow velocity and the more homogenous Mach number distribution at the rotor outlet.

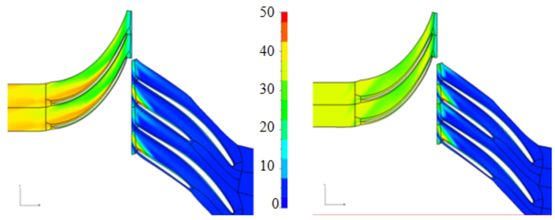

Fig. 7. Entropy [J/(kg*K)] field distribution at 10% span, for the V15 design (left) and V25 (right)

Flow field analysis near the rotor-stator interface provides further indications that the details of the flow patterns that occur in the two stage designs, V15 and V25 are more favorable to the latter. Due to the high loading of the radial section of the blade, a recirculation bubble is created - which leads to aerodynamic losses. As seen in Figure 8, this recirculation bubble forming near the tip of the rotor is eliminated by the new rotor. Improved angles of incidence for the stator vanes were also obtained, an average of 2.5 degrees difference between V25 and V15. Also, due to the higher absolute velocity magnitude, the machinery Reynolds number is higher which leads to better attachment of the flow to the stator vanes and improved diffusion efficiency [21]. Field plots showing enthalpy reveal that the outlet distribution is more uniform in V25 than in V15 which indicate that the losses, especially near the hub, are reduced. Secondary flow is also improved at the rotor outlet, in the vaneless diffuser region since the circulation necessary to compensate the through flow circulation is lower – since the through flow has lower velocity. Figure 9 depicts the baseline and optimized rotor mesh as well as the CAD and finished part.

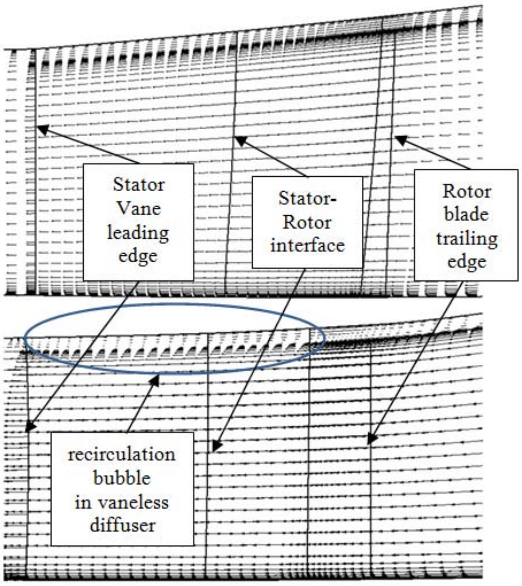

Fig. 8. Stream vectors in the vaneless diffuser of V25 (top) and V15 (bottom)

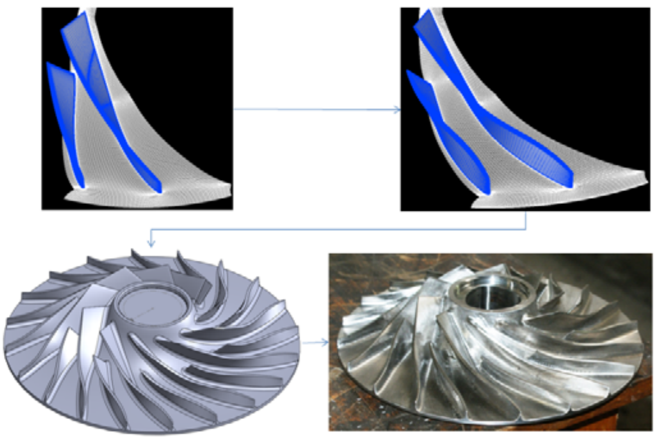

Fig. 9. The baseline (V15) and optimized (V25) rotor mesh (top), V25 CAD and finished part (bottom)

IV. CONCLUSIONS

A re-vamping and reconfiguration of an industrial HP centrifugal compressor stage was performed, using a parametric model and a combination of artificial neural networks and genetic algorithm. The design theme was to optimize the rotor geometry so that the match with the stator leads to increased stage performance and efficiency. By comparing the polytropic against the isentropic efficiencies, an apparent plateau was obtained regarding the rotor maximum efficiency. This is attributed to the mismatching of the current task (mass flow and pressure ratio) and the original diameter, wheel speed and rotor length, as confirmed by the specific speed and flow coefficient. Compared to the initial re-vamped V15 version, the V25 design obtained higher pressure ratio and improved stage isentropic efficiency. This is due to improved blade loading and lower aerodynamic losses. As shown, in V25 the rotor efficiency is influenced more heavily by the thermodynamic losses and less by aerodynamic friction with the walls - as indicated by the high polytropic efficiency. Further improvements may be achieved by optimizing the rotor-stator together. Segregated optimization has proved futile, which leads to the conclusion that the holistic interactions between the components of the stage have a significant impact on the overall results. A final remark is that although the diffuser is the region where all losses are accounted for, at least some of the aerodynamic losses from a rotor can be hidden if the polytropic efficiency is not compared to the isentropic efficiency.

Nomenclature

ANN - Artificial Neural Network
CFD - Computational Fluid Dynamics
CFL - Courant Friedrichs Lewy coefficient
GA - Genetic Algorithm
SST- Shear Stress Transport
y+ - dimensionless wall distance
k - turbulent kinetic energy
ε - turbulent kinetic energy dissipation
ω - turbulent kinetic energy specific dissipation

REFERENCES

[1] M. L. Niculescu, V. Silivestru, G. Vizitiu, S. Danaila, C. Berbente, "Theoretical and numerical investigation of centrifugal and mixed compressor impellers", 8th International Symposium on Experimental and Computational Aerothermodynamics of Internal Flows, Lyon, July, 2007

[2] L. H. Jawada, S. Abdullah, R. Zulkifli, W. M. F. W. Mahmood, "Numerical study on the effect of interaction vaned diffuser with impeller on the performance of a modified centrifugal compressor", Journal of Mechanics, Vol. 30, No. 2, pp. 113-121, 2014

[3] Numeca FINE™/Turbo v9.0, Flow Integrated Environment Theoretical Manual

[4] D. Wunsch, C. Hirsch, R. Nigro, G. Coussement, "Quantification of combined operational and geometrical uncertainties in turbo-machinery design", ASME Proceedings, Multidisciplinary Design Approaches, Optimization and Uncertainty Quantification

[5] C. Hirsch, C. Dinescu, "Uncertainty quantification and non-deterministic methodologies for CFD based design-the NODESIM-CFD project", 7th European Symposium on Aerothermodynamics, Brugge, Belgium, 2011

[6] F. Montomoli, M. Carnevale, A. D' Ammaro, M. Massini, S. Salvadori, "Limitations in turbomachinery CFD", in Springer Briefs in Applied Sciences and Technology, pp. 21-32, 2015

[7] M. R. Galvas, Fortran program for predicting off-design performance of centrifugal compressors, NASA TN D-7487, 1973

[8] V. Dragan, "Aerodynamic reconfiguration and multicriterial optimization of centrifugal compressors–a case study", INCAS BULLETIN, Vol. 6, No. 4, pp. 41 – 49, 2014

[9] B. Tang, "Orthogonal array-based latin hypercubes", Journal of the American Statistical Association, Vol. 88, No. 424, pp. 1392–1397, 1993

[10] M. Rajabi, B. Ataie-Ashtiani, H. Janssen, "Efficiency enhancement of optimized Latin hypercube sampling strategies: application to Monte Carlo uncertainty analysis and meta-modeling", Advances in Water Resources, Vol. 76, pp. 127–139, 2015

[11] FINE™/Design3D v9.0 - Optimization Kernel User Manual

[12] A. Demeulenaere, A. Purwanto, A. Ligout, F. Visser, "Design and optimization of an industrial pump: application of genetic algorithms and neural network", ASME Fluids Engineering Division Summer Meeting, Paper No. FEDSM2005-77487, pp. 1519-1527, Houston, TX, USA,Vol. 1, June 19–23, 2005

[13] S. Derakhshan, A. Mostafavi, "Optimization of GAMM Francis turbine runner", International Journal of Mechanical, Aerospace, Industrial, Mechatronic and Manufacturing Engineering, Vol. 5, No. 11, pp. 2139-2145, 2011

[14] M. V. Casey, C. J. Robinson, "A method to estimate the performance of a centrifugal compressor stage", J. Turbomach., Vol. 135, No. 2, Paper No: TURBO-11-1200, 2012

[15] C. Rodgers, "The Efficiencies of single-stage centrifugal compressors for aircraft applications", International Gas Turbine and Aeroengine Congress and Exposition Orlando, USA, June 3-6, 1991

[16] C. Xu, R. S. Amano, "Empirical Design Considerations for Industrial Centrifugal Compressors", International Journal of Rotating Machinery, Vol. 2012, Article ID: 184061, 2012

[17] R. H. Aungier, Centrifugal Compressors: A Strategy for Aerodynamic Design and Analysis, ASME Press, New York, 2000

[18] W. Al-Busaidi, P. Pilidis, "A new method for reliable performance prediction of multi-stage industrial centrifugal compressors based on stage stacking technique: Part II–New integrated model verification", Applied Thermal Engineering, Vol. 90, pp. 927–936, 2015

[19] M. Nili-Ahmadabadi, M. Durali, A. Hajilouy, "A Novel Aerodynamic Design Method for Centrifugal Compressor Impeller", Journal of Applied Fluid Mechanics, Vol. 7, No. 2, pp. 329-344, 2014

[20] P. Epple, F. Durst, A. Delgado, "A theoretical derivation of the Cordier diagram for turbomachines", Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science, Vol. 225, pp. 354-368, February 1, 2011

[21] C. Weigel, C. L, Ball, "Reynolds number effect on overall performance of a 10.8 centimeter (4.25 in) sweptback-bladed centrifugal compressor", NASA TN D-6640, 1972

AUTHORS PROFILE

Valeriu Dragan - obtained his Ph.D. in 2012 from the Politehnica University of Bucharest in the field of supercirculation aerodynamics. Since then he joined the CFD department of the National Research and Development Institute for Gas Turbines - Comoti where he specialized in centrifugal compressor design and optimization for both industrial and aeronautical applications.

Ion Malael - obtained his Ph.D. in 2013 from the Politehnica University of Bucharest in the field of flow control - with applications to turbomachineries. Since 2009 he has worked as a scientific researcher with the NRDI Comoti in a variety of CFD applications, ranging from wind turbines, cryogenic pumps to screw compressors.

Bogdan Gherman - obtained his Ph.D. in 2013 from the Politehnica University of Bucharest, following a 3 year long study stage at the Lund University, Sweeden in the field of computational aero-thermodynamics with combustion. Since 2005 he has been leading the CFD department of NRDI Comoti, in both National and European projects dealing with applications such as jet pumps or supersonic combustion chambers.