

IGV position optimization for centrifugal blower

B Gherman, M Gall, V A Popa and V-A Sterie

National Research and Development Institute for Gas Turbines COMOTI, Department of Computational Gas, Dynamics, 220D Iuliu Maniu Blvd., District 6, 061126, Bucharest, Romania

E-mail: Bogdan Gherman, bogdan.gherman@comoti.ro

Abstract: There is a need for centrifugal blowers to adapt to different operating conditions in order to meet the user requirements, in a high efficiency manner. Variable Inlet Guided Vanes (known as IGVs) are a simple solution which can lead to important energy savings. The IGV consists of wedge-shaped steel blades mounted around the inside circumference of a short length inlet pipe, with the purpose to restrict, or increase the inlet flow area, thus controlling the mass flow. The aim of this research is to determine the optimal distance between the IGV and the rotor in order to achieve maximum efficiency. The study will focus on the pressure losses and the separation of the boundary layer at the different lengths, for a fixed angle of the IGV of 45 degrees. For this study, the commercial CFD software Ansys CFX will be used to estimate the efficiency of the rotor for different set-ups, via a 3D steady-state numerical flow analysis. The end results of this study will help correlate the position of the IGV to the efficiency of the blower and to determine the optimal distance between the rotor and the IGV for an increased performance.

1. Introduction

Centrifugal compressors are at the forefront of many industries, being used in a wide area, ranging from air blowers to oil pumps. Most of the time, compressors are required to work at nominal output values for a large range of input values of the mass flow, pressure and density [1]. Given their extensive use, even a small increase in efficiency would play a big part in global energy savings. In day to day use of centrifugal compressors, large quantities of energy go to waste, due to normal irregularities of the inlet thermodynamic parameters. The Inlet Guided Vanes (IGVs) come in handy as an efficient way of adjusting these parameters, with negligible change in pressure ratio and shaft speed [11], to ensure that the compressor is working at nominal parameters.

IGVs play an important role in optimizing the use of energy in a compressor/blower, allowing power savings of up to 9% to the compression system [2]. They do so by restricting the flow of fluid to the desired/nominal value, thus saving the energy that would otherwise be used to compress an unnecessary amount of fluid. As an alternative to the conventional inlet butterfly valve, an IGV is mounted directly to the inlet flange of the compressor and it has multiples triangular aerodynamically profiled blades. These blades are variable and offer minimum restriction in the full open position, but they can also be operated fully closed in order to achieve an efficient compressor unloading [3].

Several studies were conducted in order to improve the way IGVs are integrated in compressors/blowers in order to reduce the amount of energy used for the compression process. Experimental research on a centrifugal rotor equipped with two different types of IGVs, differentiated by geometric parameters of the airfoil, revealed that energy consumption can be reduced by about 2%-



3% at the same flow rate when the vane angle varies from 50° to 90° [4]. The first improvement method proposed by this paper consisted in a dual linear segment of the airfoil shape which led to a reduction of the drag coefficient at the same angle of the vane and at the same time an increase of the pre-rotation coefficient was observed. The second improvement was the increment of cascade solidity at small radii. This solution showed an obvious greater pre-rotation coefficient, while, surprisingly, the drag coefficient remained almost the same.

Another experimental study in the field of inlet guide vanes was conducted by Zhang et al. [5]. Using a probe with five holes, the upstream and downstream flow fields were measured for a centrifugal rotor equipped with a straight bladed IGV. As a result, the IGV was proved to not have significant influence on the upstream flow distribution. Moreover, the study concluded that the increment of the IGV's pitch angles led to a surge in the drag coefficient and the pre-rotation coefficient. Furthermore, an optimal gap between the IGV and the impeller was determined at around 4-8 times the length of the blade chord. The authors also suggested some possible future proposals to enhance the performance of the IGV, by modifying the airfoil shape, and a study on optimization of the cascade solidity.

Fukutomi and Nakamura also put their efforts into studying the effect of angle and length of an inlet guide vane on the performance of a cross-flow fan [6]. They reached the conclusion that the fan equipped with the IGV performed better thus obtaining a higher pressure and also a higher efficiency.

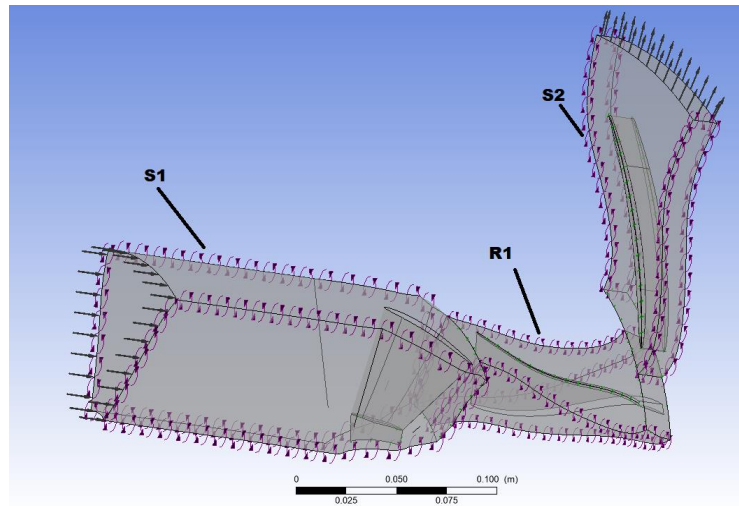
An extensive study on performance optimization for different inlet conditions has been made by Ling and Sönne [7]. In this study, it was shown that the inlet temperature has a big influence on the efficiency of turbomachinery and guided vanes can greatly counteract the negative influence of undesired condition. Six cases have been conducted in this study. In the first one, the performance was evaluated by modifying only the shaft's speed, the second one showed the influence in performance for different ambient temperatures, while in the third one, showed the most suitable way of controlling the guided vanes. The fourth study examined the most promising positions from the first three studies and the last two are related to the interaction between the SGT-700 engine and the previous cases. The study was conducted by numerical simulations, complemented by the supervision of Siemens Industrial Turbomachinery. This study concluded that the overall power can be enhanced by almost 3% for a turboprop if the guide vane position is correctly correlated with the inlet conditions and this can lead to the idea of implementing guide vanes to other types of machinery such as compressors, blowers or fans.

Another extensive experimental study on turbulent structures in a low speed axial fan with IGVs has been made by Jesús Manuel Fernández Oro et al. [12]. In this study a complete picture of turbulent flow has been determined using hot-wire anemometry in different operating points.

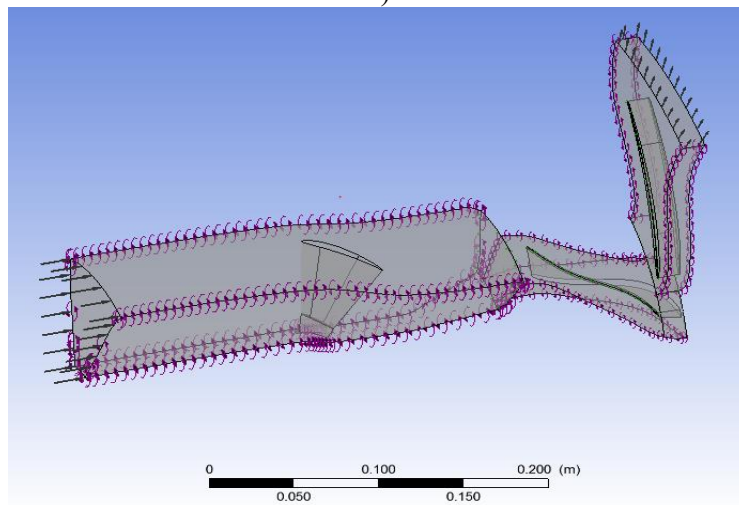
Numerical simulations along with subsequent experimental validation have been also implemented in order to determine the performance characteristics of an existing IGV [8]. These numerical simulations led to the development of a new vane and also, improvements regarding the inlet ducting geometry were made. Numerical investigations were also undergone in order to analyze the influence of an inlet guide vane to the unsteady flow of a centrifugal pump [9]. Moreover, comparisons between experimental and numerical results led to an optimal choice regarding the mesh elements, time steps and turbulence models.

The inlet guide vanes can also be beneficial to the performance of smaller fans. With the fast development of electronic industry towards the reduction of dimensions and increase in power, the performance of small cooling fans needs more and more attention. A study of those benefits is made by LIU Yang et al. [10] for a small axial flow fan, used for cooling electronic devices. In this study both numerical and experimental analysis are made and the results show that an IGV can drastically increase the flow performances and can reduce noise if well positioned. The simulation was computed for both steady and unsteady states. For the steady state, Navier-Stokes equations coupled with RNG k-epsilon turbulent model were used, while the unsteady flow was computed with large eddy simulation (LES).

The vane profiles are responsible for the aerodynamic stability which limits the efficient operation of the IGV. Three variants of vane profiles (symmetric profile – used as reference, two-piece tandem profile, and a s-cambered profile) were tested both numerically and experimentally by A. Mohseni et al. [11] at different negative and positive pitch angles and at different operating points.



a)



b)

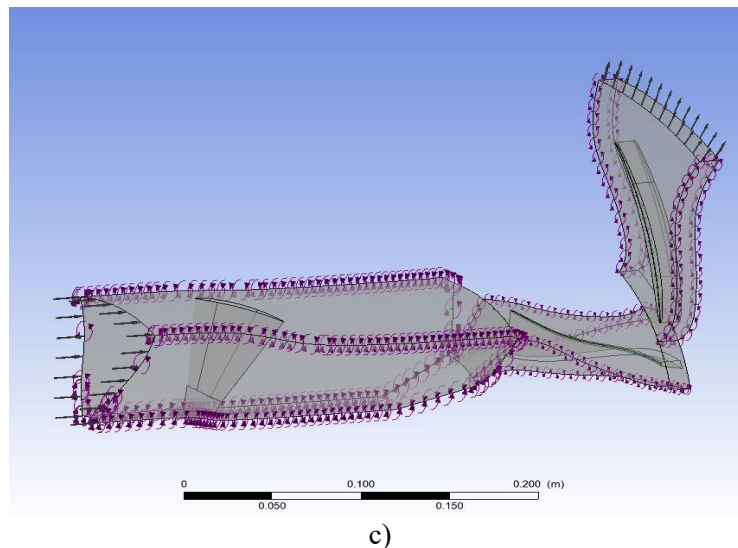


Figure 1 a) Domain distribution CASE 1; b) Domain distribution CASE 2;
c) Domain distribution CASE 3.

It was concluded that tandem and s-cambered profiles have superior aerodynamic performance and offer the possibility to extend the high efficiency operating range of the compressor. Furthermore, when it comes to costs and manufacturing, the s-cambered profile proved to be the most suitable for positive pre-swirl cases of operation.

As already stated, there are some optimization criteria which can be studied when an IGV is to be mounted on a certain compressor (number of blades, type of blade airfoil, axial distance between the IGV and the rotor). The aim of this research is to determine the influence of the axial distance between the IGV and the rotor to the overall efficiency of the compressor by studying the losses of pressure, the separation of the boundary layer at different lengths and the whirling pattern induced to the flow. All the analysis will be made using numerical simulations with commercial software for a fixed angle of 45 degrees for the IGV. The final purpose is to determine the optimal axial distance for the IGV to be placed. Future plans include experimental testing and validation of the results obtained in this paper with a view to determine the return period of such an investment for a compressor.

2. Case setup

2.1 Domain details

The computational domain consists of three regions, which are: S1-the domain representing the inlet duct containing the IGV; R1-the centrifugal rotor; S2-the stator domain. Three types of configurations were simulated. Each case has a different axial distance between the rotor and the IGV blade. In the first simulation, the IGV is very close to the rotor (approximately 3mm) (figure 1 a.), while for the next two cases the distance increases by 75mm (figure 1 b.), and 150mm (figure 1 c.), respectively. The last two distances were chosen by taking the mean chord of the IGV profile (which is approximately 25mm) and multiplying it by 3 and by 6 respectively.

2.2 Mesh

For the IGV domain (S1), the mesh was computed in ICEM CFD (figure 2 a)), as a structured grid using the blocking function. The outer side of the blade was created using the O-grid function with the purpose to improve the overall mesh quality and to have a better control of the phenomena in the vicinity of the wall (figure 2b). The cell size near the wall is $10^{-6}m$ for a y^+ value of 1.

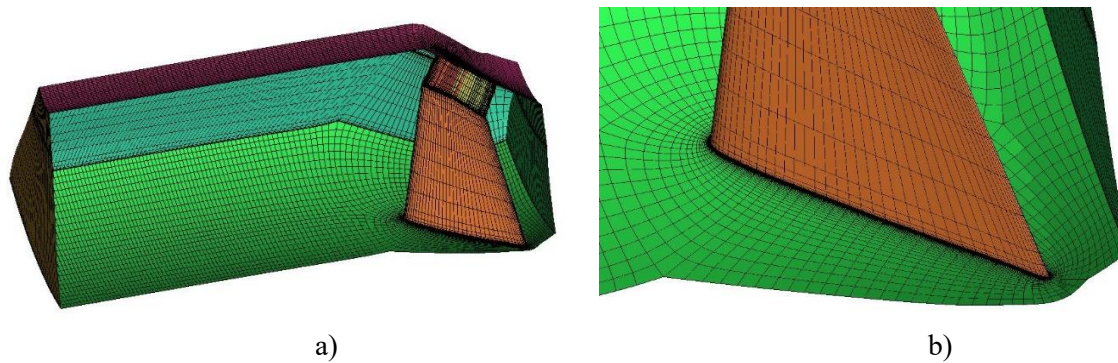


Figure 2. a) S1 mesh; b) S1 mesh detail.

Both R1 and S2 domains have meshes computed in TurboGrid. For R1 (figure 3 a.) the normal distance between the shroud and the blade has been set to 0.5 mm (figure 3 b.), and the mesh has been thickened with 15 radial elements in this zone. Also, as the blade profile has a cutoff trailing edge, the number of elements in this area has been tripled for a more accurate solution (figure 3 c.). For S2, the normal distance between the blade and both the shroud and the hub has been set to 0.5 mm and 15 elements have been added there too.

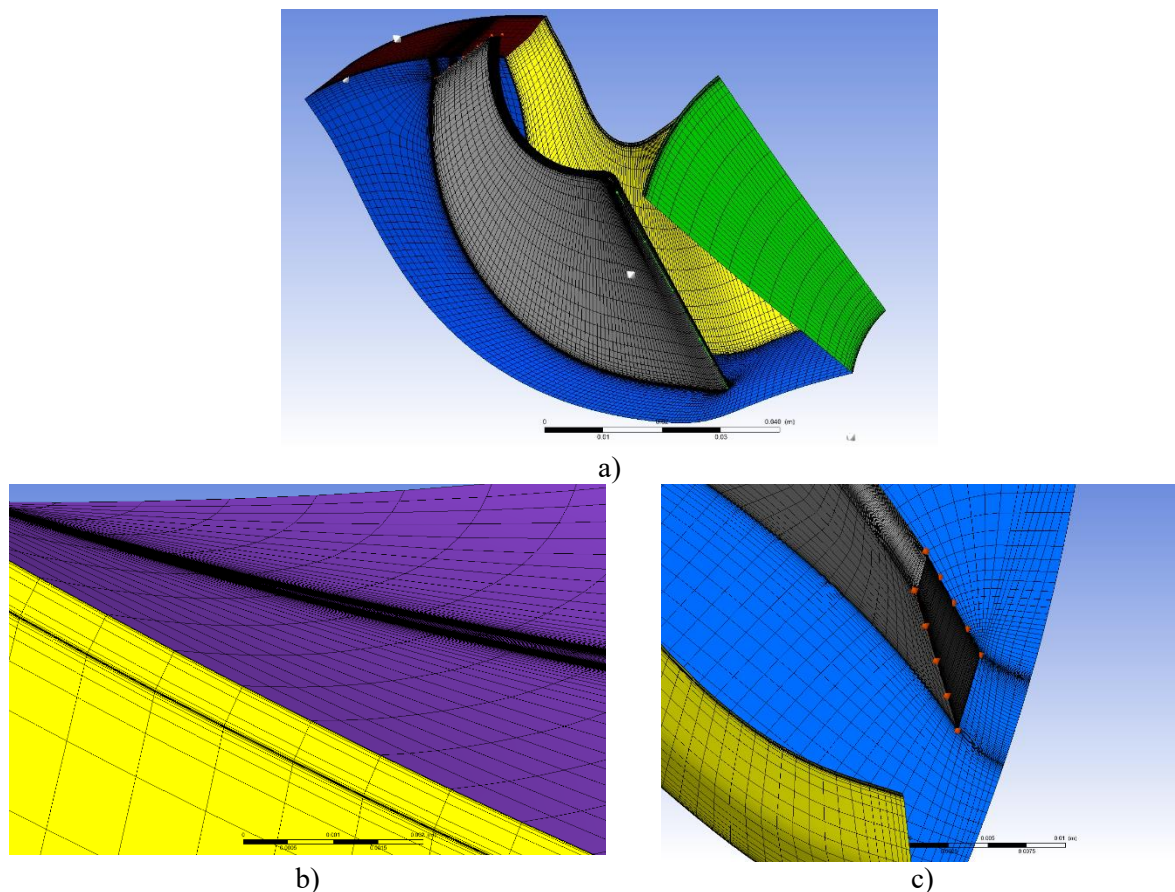


Figure 3. a) R1 mesh; b) R1 mesh at shroud tip; c) R1 mesh at trailing edge.

The figures listed in table 1. are for the case where the number of cells was minimal, i.e. the case where the IGV is the closest to the rotor. The other two cases have a greater number of cells because of the larger axial distance.

Table 1. Mesh characteristics.

Domain	Number of cells
Stator 1 (S1)	469890
Rotor 1 (R1)	307756
Stator 2 (S2)	499540
Total	1277186

2.3 Boundary conditions

The boundary conditions were all set in CFX-Pre, after the three parts of the computational domain have been put together.

The IGV blade, the centrifugal rotor blade, the hub and the shroud for the three components (S1, R1, S2) are defined as smooth walls with no slip condition and adiabatic heat transfer. The periodics were defined as interface with conservative interface flux for mass and momentum, turbulence and heat transfer.

The rotor domain has a rotating domain motion given by an angular velocity of 22784 rot/min.

The reference pressure is 1 bar. As a result, the inlet relative pressure is set to 0 bar, whereas the outlet relative pressure is set to 0.5 bar, thus the simulation performed is pressure-inlet pressure-outlet type.

2.4 Turbulence model

The main problem in turbulence modeling is the accurate flow prediction of the flow separation from a smooth surface. This is a very important phenomenon, especially in aerodynamics, because the performance of different airfoils can be overly optimistic simulated if standard two-equation turbulence models are used. In the same way, in turbomachinery CFD calculation, the stall can be overlooked if using standard turbulence models.

For these reasons, some advanced turbulence models were developed. Currently, the most used two-equation model in this area, is the SST $k - \omega$ model. This model is designed to give accurate predictions of the onset and the amount of flow separation under adverse pressure gradients by the inclusion of transport effects into the formulation of the eddy-viscosity. Moreover, this model switches to $k - \varepsilon$ behavior in the free-stream and in this way the $k - \omega$ sensitivity to inlet free stream turbulence properties is avoided. Another advantage of the $k - \omega$ is the accuracy and robustness for low-Reynolds numbers flows, near the walls [13].

In this study, the SST (Shear Stress Transport) $k - \omega$ turbulence model was used. This model is based on two transport equations, one for the turbulence kinetic energy:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_j k)}{\partial x_j} = P - \beta^* \rho \omega k + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_j} \right] \quad (1)$$

and the other for the specific dissipation rate:

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho u_j \omega)}{\partial x_j} = \frac{\gamma}{v_t} P - \beta \rho \omega^2 + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_j} \right] + 2(1 - F_1) \frac{\rho \sigma_{\omega 2}}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \quad (2)$$

In those equations the eddy viscosity μ_t is defined as:

$$\mu_t = \rho \frac{k}{\omega} \quad (3)$$

The closure coefficients of the SST k - ω model are:

$$\alpha = \frac{5}{9} \quad \beta = \frac{3}{40} \quad \beta^* = \frac{9}{100} \quad \sigma = \frac{1}{2} \quad \sigma^* = \frac{1}{2} \quad (4)$$

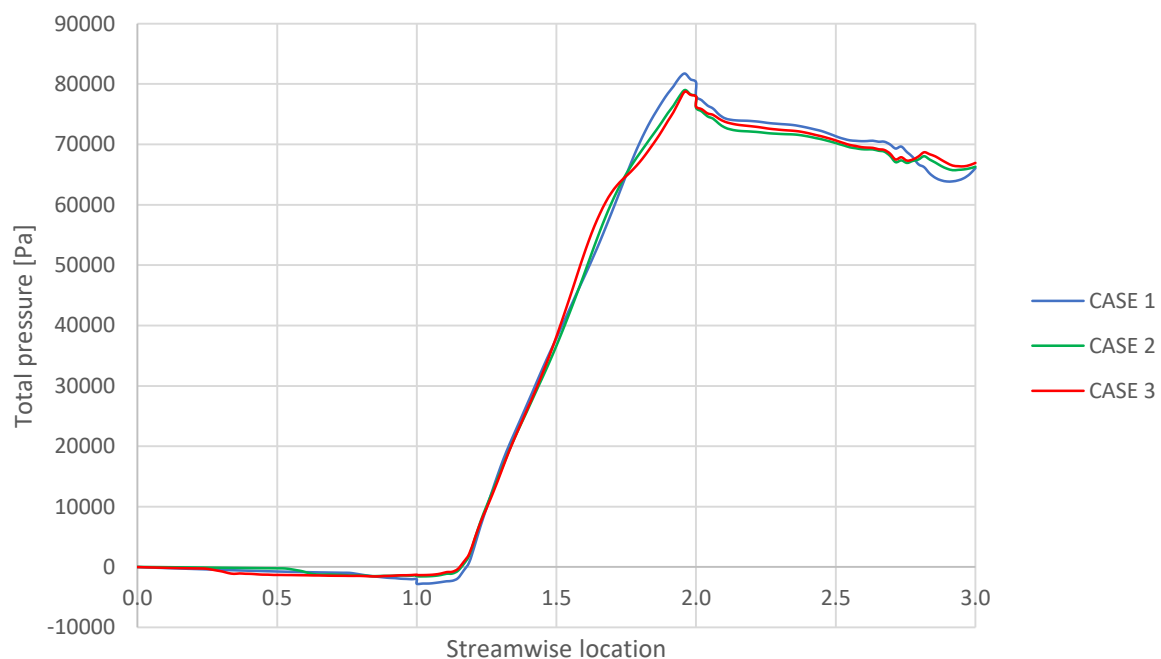
And other two important relations which are used:

$$\varepsilon = \beta^* \omega k \text{ and } l = \frac{k^{0.5}}{\omega} \quad (5)$$

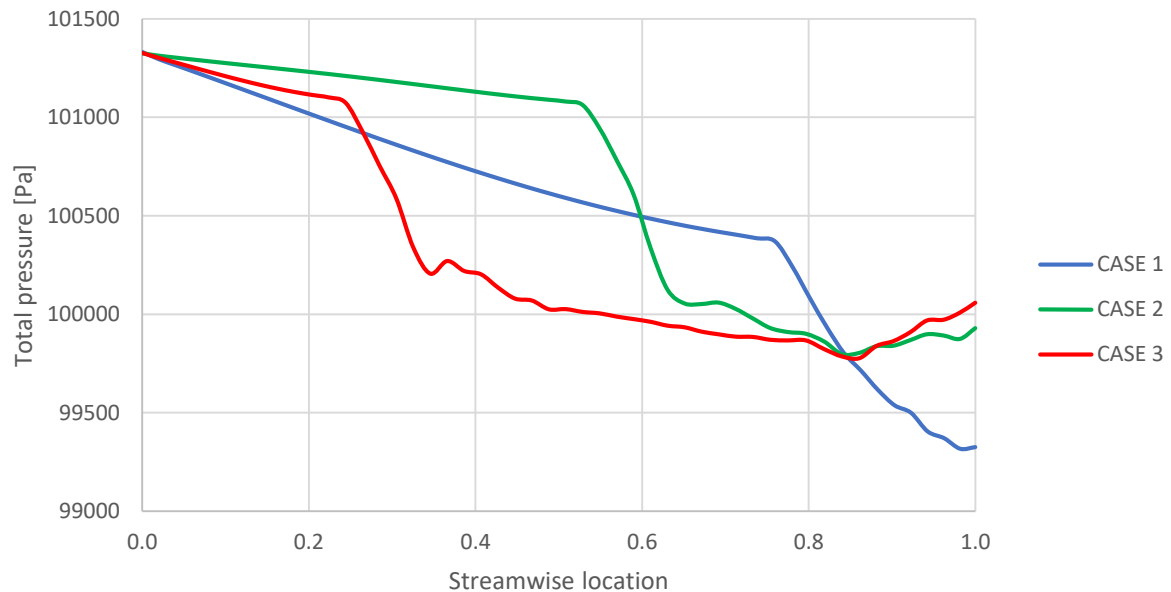
3. Results and discussions

In the paragraphs below, the different cases will be referred to as follows: CASE 1 - the IGV is very close to the rotor (approximately 3 mm), CASE 2 – the axial distance between the IGV and the rotor is 75 mm, and CASE 3 - the axial distance between the IGV and the rotor is 150 mm.

The total pressure variation along the centrifugal blower after post-processing and the results from the simulation are plotted in figure 4 a). At first glance, a slight decrease in total pressure can be noticed before entering the rotor (pressure loss). It can also be seen that there are differences regarding the point where the pressure loss becomes noticeable. These differences correspond with the change in geometry from one case to another. Figure 4 b) offers a closer look to the total pressure values for the inlet domain. As the axial distance between the IGV blade and the rotor increases, the start point for pressure losses moves towards the inlet. The maximum pressure loss was recorded for CASE 1 when the total pressure decreased to 99400 Pa, whereas CASE 2&3 reached a value of about 100000 Pa after flowing through the inlet via IGV blades. The same pressure loss phenomenon is present in the stator and leads to a decrease in total pressure which is quite similar for the three cases studied in this paper (figure 4 a). At a closer look at this figure, a total pressure of approximately 1.8 bar is obtained after the compression in the rotor for CASE 1, whereas the outlet pressure decreases to 1.65 bar. For CASE 2&3 the total pressure after the compression in the rotor is a little bit different and it reached about 1.78 bar.



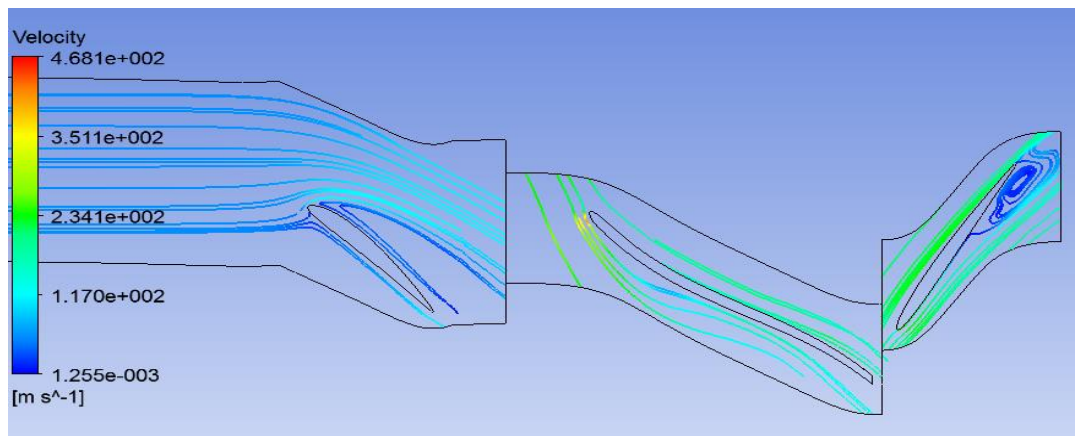
a)



b)

Figure 4. a) Total pressure variation along the centrifugal blower; b) Total pressure variation along the inlet domains.

A blade to blade graph with the streamlines can be seen in figure 5. As the angle of attack of the IGV blade is at 45° , an early boundary layer separation can be observed which results in a high vortical pattern before entering the rotor. It can be seen that, by increasing the distance between the vane and the rotor, the vorticity magnitude near the inlet is greatly reduced, which might account to the increase in overall efficiency. A recirculation bubble also appears in the stator, but there are no visible differences between the three cases.



a)

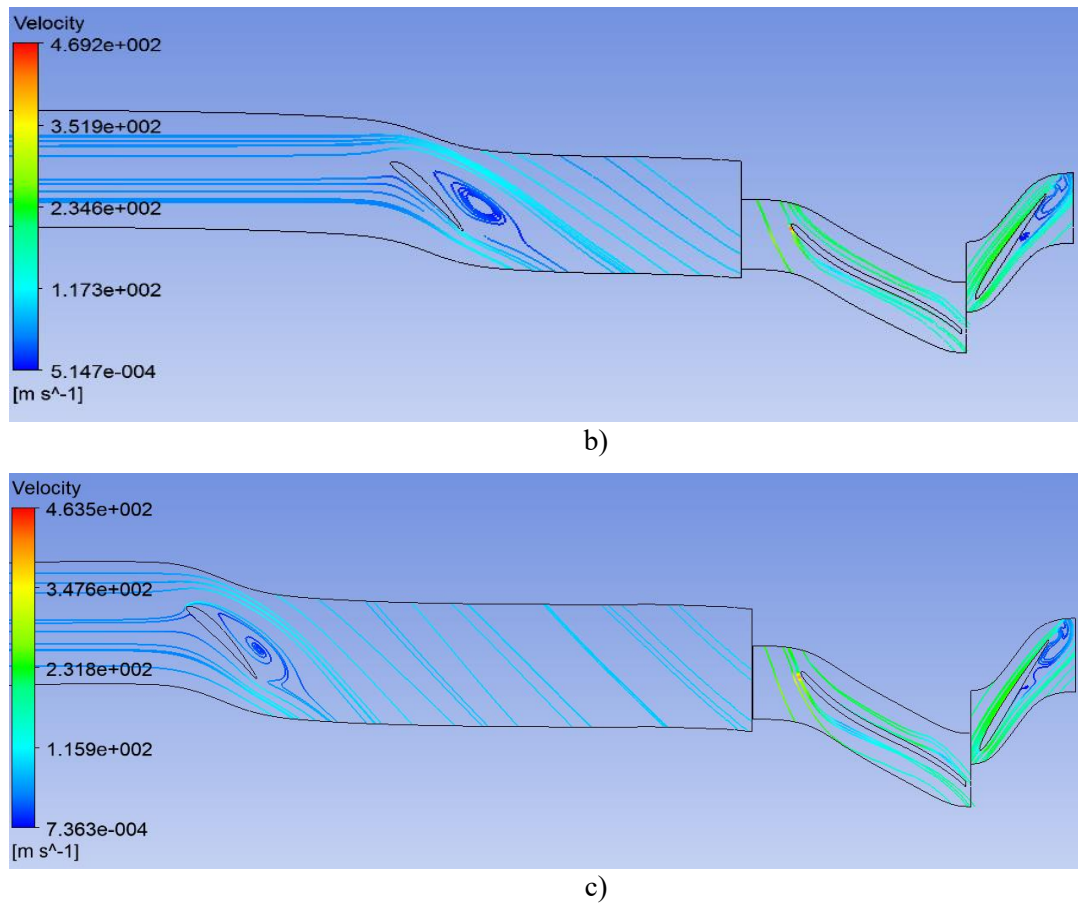
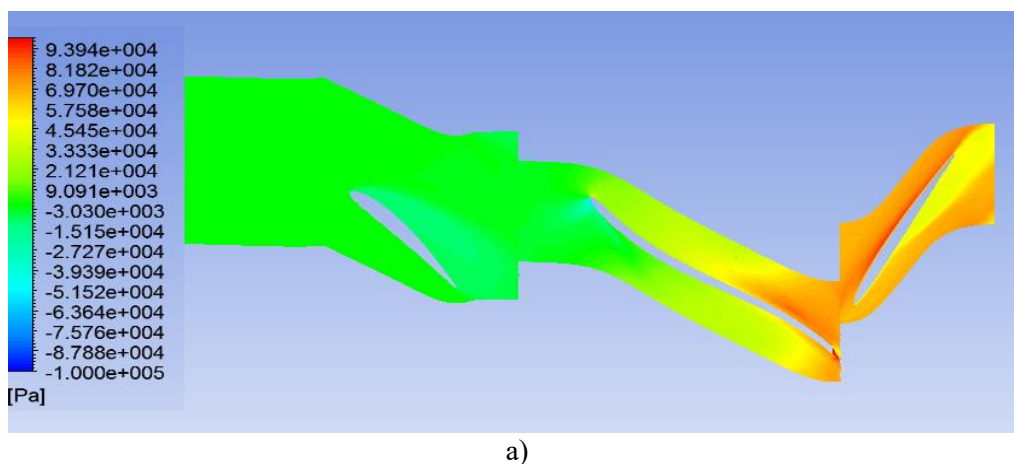


Figure 5. a) Streamlines (blade to blade view) – CASE 1; b) Streamlines – CASE 2; c) Streamlines – CASE 3.

A blade to blade view with the total pressure variation is shown in figure 6. In this figure, it can be observed that the zones with the maximum pressure losses are strictly correlated to the recirculation zones. Consequently, the maximum drop in pressure is marked at the pressure side of the stator blade, a zone in which we can also observe a high vorticity pattern for all the three cases. At a closer look, pressure losses exist at the suction side of the IGV blade too. Figure 5, also shows zones of recirculation in that area.



a)

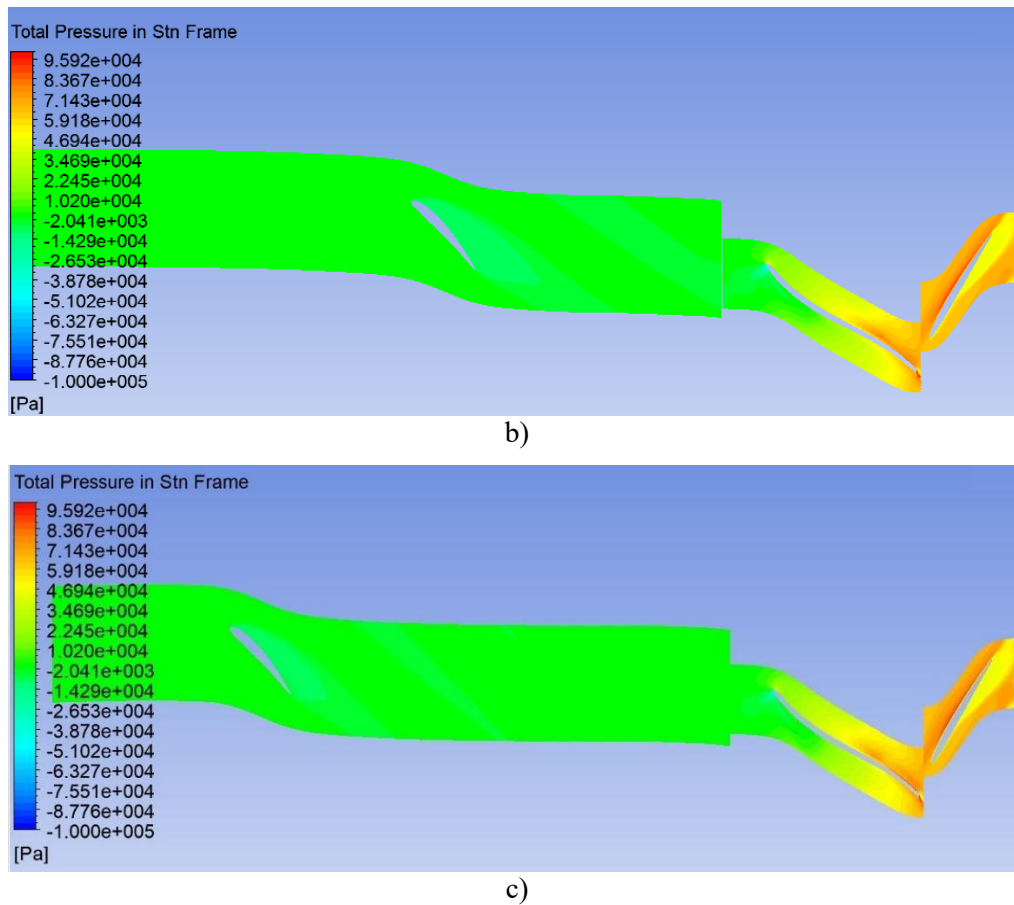


Figure 6. a) Total pressure variation – CASE 1; b) Total pressure variation – CASE 2; c) Total pressure variation – CASE 3.

The results for each simulation show that indeed, by placing the IGV further apart from the rotor, the overall polytropic efficiency increases. At the same time, the growth rate of the efficiency decreases with the increase in distance. This observation is given by the results, which show that, by placing the IGV at 75 mm apart from the rotor, the overall efficiency increases by 1.14%, while placing the IGV at 150 mm, the performance only increases by a mere 0.1%.

Table 2 shows the results of CFX-Post calculation for overall polytropic efficiency for each case.

Table 2. Overall efficiency for each case.

Case #	Efficiency
Case 1 (3 mm)	0.714756
Case 2 (75 mm)	0.726213
Case 3 (150 mm)	0.727701

The performance augmentation brought by the increased distance, might be caused by the smoother streamlines. As shown in figures 5 a, b, c, this smoothening happens without modifying the direction of flow given by the IGV, direction which is needed at the rotor inlet for a more efficient compression.

4. Conclusions

The conclusion is that the optimization of IGVs depends on the practical application in which the compressor/blower is used. If the application demands a constant use of the system, then an increase

of 1% in efficiency can bring great benefits in energy savings. On the other hand, if a more robust geometry is needed, the distance between IGV and the compressor can be shortened, taking in consideration a maximum distance possible for a practical use. Another aspect that should be considered, is the fact that the angle of attack of the IGV (45 deg.), is a rare occurrence for a compressor, most of the time, the angle varies between smaller values, that would enable a more stable flow, with low vorticity magnitudes throughout the first domain. Further studies that show the effect on the efficiency of both the angle of attack of the IGV and the distance between the IGV and the rotor will be conducted in order to obtain an optimal solution for the system.

5. References

- [1] Coppinger M and Swain E 1999 Performance Prediction and Optimization of an Industrial Centrifugal Compressor Inlet Guide Vane System, *Proceedings of the European Turbomachinery Conference*, London, UK, Paper No. C55/065/99
- [2] <https://www.airbestpractices.com/technology/air-compressors/variable-inlet-guide-vanes-boost-centrifugal-air-compressor-efficiency>
- [3] <https://www.ingersollrandproducts.com/en-ca/parts/compressed-air-parts-and-accessories/upgrade-variable-inlet-guide-vanes.html>
- [4] Zhang Y, Qi D, Tan J and Mao Y 2008 *Experimental Study on Performance of Inlet Guide Vane with Improvement of Blade Airfoil and Cascade Solidity* May 2008
- [5] Zhang Y, Qi D, Tan J and Mao Y 2007 *Experimental study on inlet guide vane with straight blade in a centrifugal fan* September 2007
- [6] Fukutomi J and Nakamura R 2005 Performance and Internal Flow of Cross-Flow Fan with Inlet Guide Vane *JSME International Journal Series B Fluids and Thermal Engineering*, ISSN : 1347-5371
- [7] Ling S and Sönne T 2014 *VIGV optimization for performance* (Thesis for the Degree of Master of Science Division of Thermal Power Engineering Department of Energy Sciences)
- [8] Coppinger M and Swain E 2000 Performance prediction of an industrial centrifugal compressor inlet guide vane system *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*
- [9] Yuchuan W, Lei T, Baoshan Z, ShuLiang C and Binbin W 2015 Numerical investigation of influence of inlet guide vanes on unsteady flow in a centrifugal pump **229**(18)
- [10] Yang L, Zhe L, Peifeng L, Yingzi J, Setoguchi T and Kim H D 2017 Effect of Inlet Guide Vanes on the Performance of Small Axial Flow Fan *Science Press and Institute of Engineering Thermophysics* CAS and Springer-Verlag Berlin Heidelberg
- [11] Mohseni A, Goldhahn E, Van den Braembussche R A and Seume J R 2011 Novel IGV Designs for Centrifugal Compressors and Their Interaction with the Impeller *ASME. J. Turbomach* **134**(2)
- [12] Manuel J, Oro F, María K, Díaz A, Santolaria Morros C and Marigorta E B 2007 On the structure of turbulence of a low-speed axial fan with inlet guide vanes *Experimental Thermal and Fluid Science* Universidad de Oviedo, Área de Mecánica de Fluidos, Edificio Departamental Zona Este, Campus de Viesques, Spain.
- [13] *ANSYS Documentation>CFX>Modeling Guide>Turbulence and Near-Wall Modeling>4.1.5. The k-omega and SST Models.*