

Numerical and experimental evaluation of a centrifugal compressor

B Gherman, O Dumitrescu and M Nitulescu

National Research and Development Institute for Gas Turbines COMOTI, 220 D Iuliu Maniu Bd., sector 6

E-mail: bogdan.gherman@comoti.ro

Abstract. The present paper provides a comparative study in the development of a centrifugal compressor impeller designed for methane (CH_4) and tested in similitude conditions, using air as working fluid. The rotor was designed in maximum efficiency conditions and will be part of compression equipment used for underground gas storage for inaccessible areas, experiencing difficulties in gas supply. Optimization of the geometrical shape of the impeller was made using CFD programs based on the finite volume method. Since the experimental tests have taken place at the similitude, the rotating speed had to be adapted to 14915 rpm, to compensate for the change in working fluid. Redundant pressure and temperature probes have been placed on each radial location in order to minimize experimental errors. For the CFD side, the established best practices were employed, using the k- ω SST model with a $y^+ < 1$ along the walls, with a growth ratio of $\sim 1.2:1$. Overall, the global performances were in good agreement with each other

1. Introduction

In this paper, the development of centrifugal compressors for industrial natural gas transport and storage at high pressures is presented.

Centrifugal compressors are one of the most robust machineries used as effective solutions for many problems related to the transportation and compression of gases. The relatively small dimensions and high compression ratios, with high overall efficiencies, are the main advantages of centrifugal compressors [1].

In addition to power plant and propulsion applications, they may also provide a solution for the storage of methane (CH_4) in inaccessible areas experiencing where gas supplying is difficult. In this case, a high performing turbo-machinery is needed in order to increase storage capacities, and to ensure the safety of gas supply. Also, since gas compression is a very energy intensive task, it is essential to operate in the most efficient way possible, so that resources are used at their full capacity [2]. Having in sight the safety factor, the methane stored must not exceed specific prescribed temperatures, at different pressures [3], so that the auto-ignition can be avoided.

Since detailed experimental data provides exact information under strict laboratory conditions, it is possible to map the impeller performances with this data, but only for research purpose. On the other hand, for industrial needs, other methodologies must be implemented. Fortunately, it has been proven [4] that there are several safe methods of transforming the compressor's performance curves, one of which is the similitude conversion principle. The similitude conversion of centrifugal compressors is



based on the affinity law; if the processes of the model and prototype are similar, the ratios of performance parameters are equal, and so are the efficiency and the loss parameters [4].

One output of the experimental testing is the possibility to compare the data against the CFD methodology, hence calibrating it as a method. Turbulence model accuracy is influenced, amongst other factors, by the operating conditions [5]. Furthermore, the compressor characteristics of different inlet conditions could be given by computation, using commercial software (in the current case, ANSYS CFX) in which the SST k- ω (with compensation for rotation and curvature) could be the turbulence model. It has been shown that this model gives accurate results [6-9], even in cases where minute differences need to be discriminated, such as optimization studies or cases where heat transfer is relevant. In the process of developing impellers with the high isentropic efficiency; both experimental and numerical data must be involved. For the experimental part, the instrumentation of the compressor must have a complete set of measurement tools, correctly placed, so that they can provide accurate information on the parameters needed to be studied. A study of an effective test rig for centrifugal compressors has been made in [10].

Numerical CFD analysis permits to validate a new geometry, without the need for prototyping and labour intensive experimental campaigns. It also provides an image of the fluid flow within the work assembly, allowing the analysis and understanding of the qualitative phenomena taking place in the compressor [11]. Also, this type of analysis is difficult to obtain on a test rig, due to its complex nature [12].

2. Experimental and numerical setup

For safety reasons, the experimental tests are made for a centrifugal compressor, tested in similitude conditions with air rather than methane gas (for which the compressor was ultimately designed). The impeller was tested in maximum efficiency conditions and will be part of compression equipment used for underground gas storage in areas experiencing difficulties in gas supply. The goals are to increase the storage capacity of existing facilities while maintaining strict gas supply safety;

An exterior diameter of 350 mm and 21 blades were obtained for a mass flow of 0.437 kg/s. The impeller was executed on the machine in 5 axes, the design data being transmitted to the CAD-CAM system. The impeller was tested at the similitude rotating speed: 14915 rpm.

Figure 1 illustrates the working conditions for the centrifugal compressor stage, presenting the following conditions: 2 – Compressor inlet; 3 – Impeller inlet; 4 – Impeller outlet; 5 – Diffuser inlet; 6 – Vaned diffuser inlet; 7 – Vaned diffuser outlet; 8 – Diffuser outlet; 9 – Compressor exit.

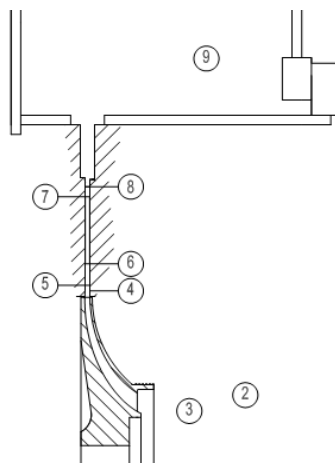


Figure 1. Working area of the compressor stage.

An experimental program has been defined for the conditions in which the tests could be carried out, using the similitude criteria. Figure 2 presents the impeller and the compressor equipment used for

the experimental tests, also the pressure probes used for the compressor instrumentation. For the impeller a number of 8 pressure probes have been used: 3 on the impeller inlet and 5 on the impeller exit. In this component all the probes are for the static pressure.



Figure 2. a) Impeller assembly; b) Compressor equipment.

For the vanned diffuser, both static and total pressure has been measured. Figure 3 presents the pressure probe used for this instrumentation. Total pressure has been measured using just one sample for the inlet and one for the diffuser exit. Regarding static pressure, the number of pressure probes has increased, 12 probes, and their distribution on the diffuser is presented in figure 3.a.

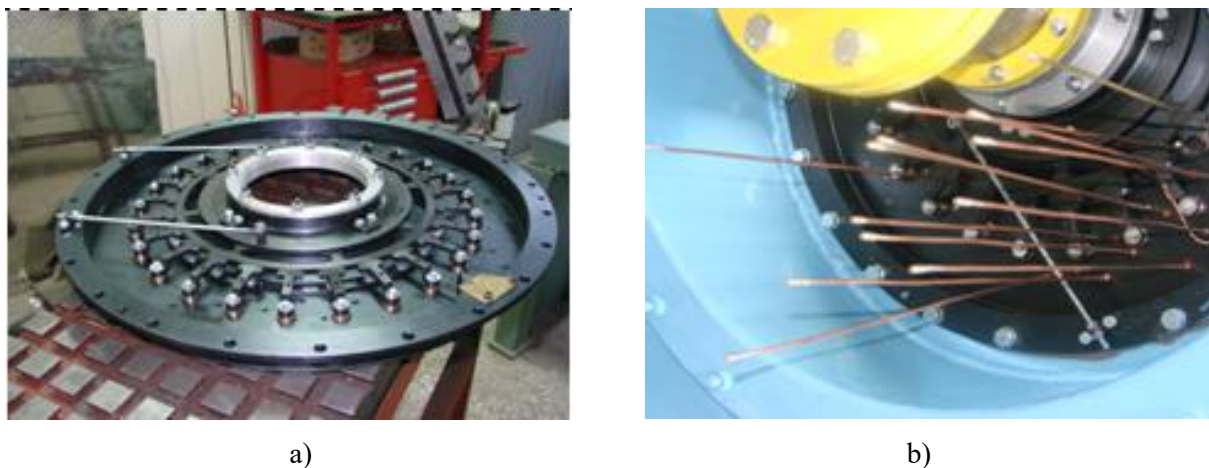


Figure 3. a) Vaned diffuser (control system); b) Detailed of pressure probes used for instrument the vanned diffuser.

Also, figure 4 illustrates the exact distribution of this pressure measurement points. The corresponding indices indicate: P6 – total pressure on vanned diffuser inlet; PS3÷14 – static pressure on diffuser; P7 – total pressure on vanned diffuser outlet.

The main features of the centrifugal compressor are presented in table 1.

Table 1. Main parameters of the centrifugal impeller.

	Dimension
Impeller diameter	350 [mm]
Diffuser exit diameter	560 [mm]
Number of blades – impeller	21
Number of blades – diffuser	25
Angular velocity	14915 [rpm]
Mass flow	0.437 [kg/s]
Total pressure on impeller inlet	0.9995 [bar]
Total temperature on impeller inlet	27 [°C]
Pressure ratio	1.7

For the similitude point the following conditions must be accomplished:

$$a) \quad \left(\frac{q_3}{N} \right)_t = [0.00143 \div 0.00155] \text{ [m}^3/\text{rot}] \quad (1)$$

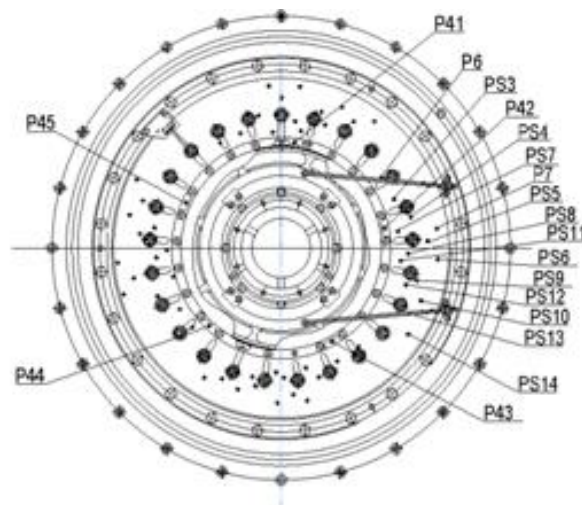
$$b) \quad \left(\frac{q_3}{q_9} \right)_t = [1.406 \div 1.554] \quad (2)$$

Where: q_3 – volumetric mass flow at the impeller inlet; q_9 – volumetric flow rate at the compressor exit; N – rotating speed; t – test conditions

In addition, it is checked the function fitting of the envelope, figure 5.

$$c) \quad \frac{(Re_m)_t}{(Re_m)_{sp}} = f((Re_m)_{sp}) \quad (3)$$

$$d) \quad (M_m)_t - (M_m)_{sp} = f((M_m)_{sp}) \quad (4)$$

**Figure 4.** Pressure measurement point in the diffuser.

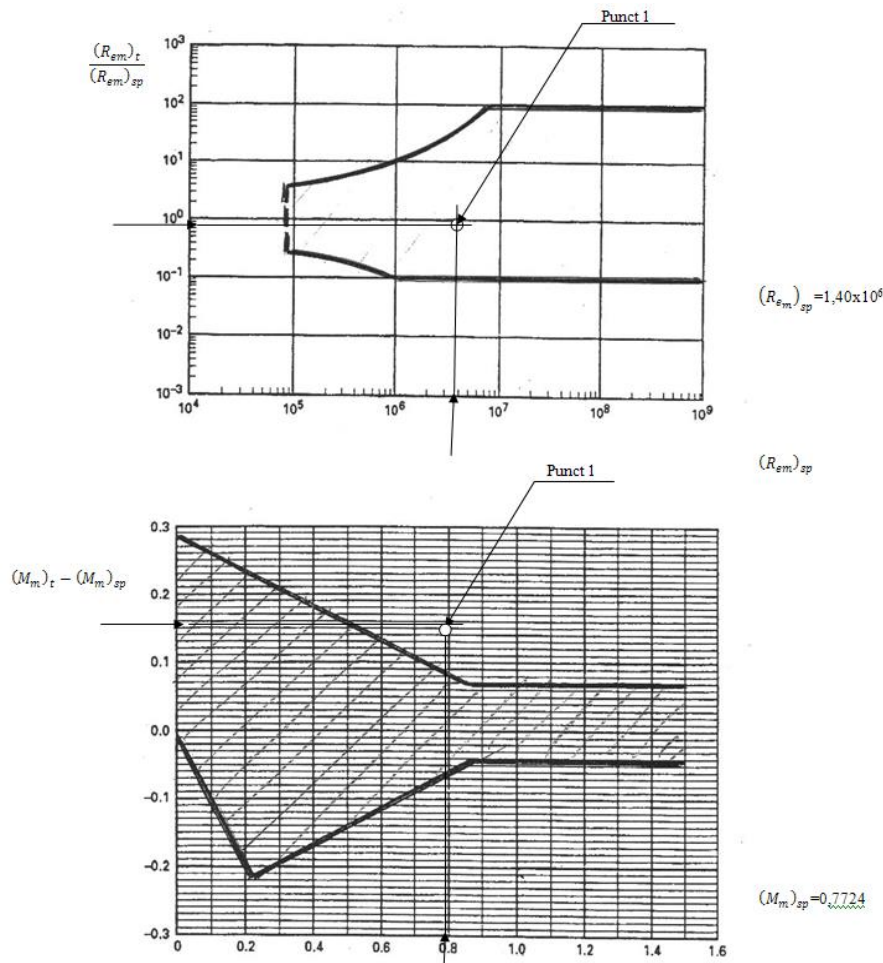


Figure 5. Determination of the similitude point.

The experimental results will be compared with the numerical ones, obtained using the commercial software ANSYS CFX. As turbulence model used for this analysis is the SST k- ω , with compensation for rotation and curvature [13], as implemented in ANSYS CFX. This model has been shown to yield accurate results, and is suitable for cases where it is important to capture both the phenomena occurring near the walls and those of the entire computing field [14].

The mesh for both components is structured, making the best use of the cell count, especially near the blade. The entire channel, impeller – diffuser, has a number of near 1.8 million elements.

3. Results

The CFD results obtained base on the numerical analysis conducted for similitude conditions, are presented below. For the given mass flow, 0.437 kg/s, a pressure ratio of 1.7:1 was obtained. Total pressure distribution on one passage, figure 6, reveals a uniform evolution of pressure.

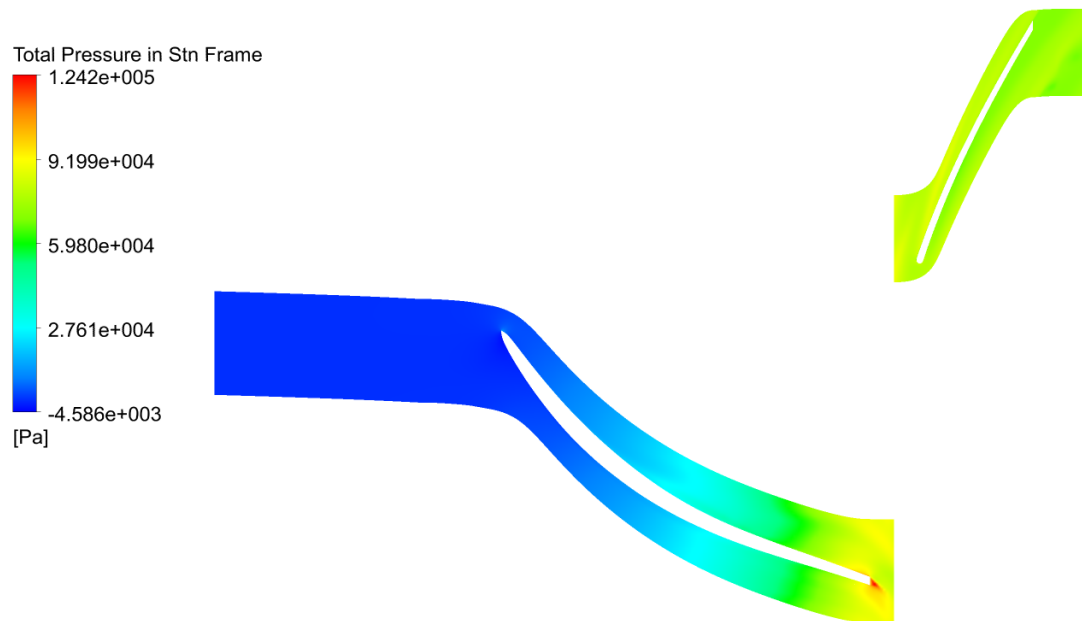


Figure 6. Total Pressure in Stn Frame 50% span.

Total pressure, meridional view, figure 7, presents a higher pressure on the impeller – diffuser interface, given by the fluid compression. As the fluid approaches the diffuser outlet, the pressure distribution is uniform.

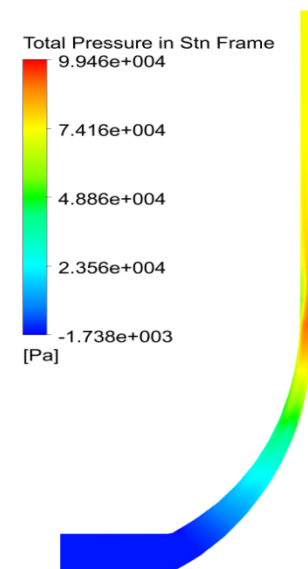


Figure 7. Total Pressure in Stn Frame, circumferentially averaged view.

Also in the case of the total temperature, figure 8, on the back of the trailing edge of the impeller blade, the value is higher and is given by this blade to blade warp form of the trailing edge, but also due to a small attached vortex in this area.

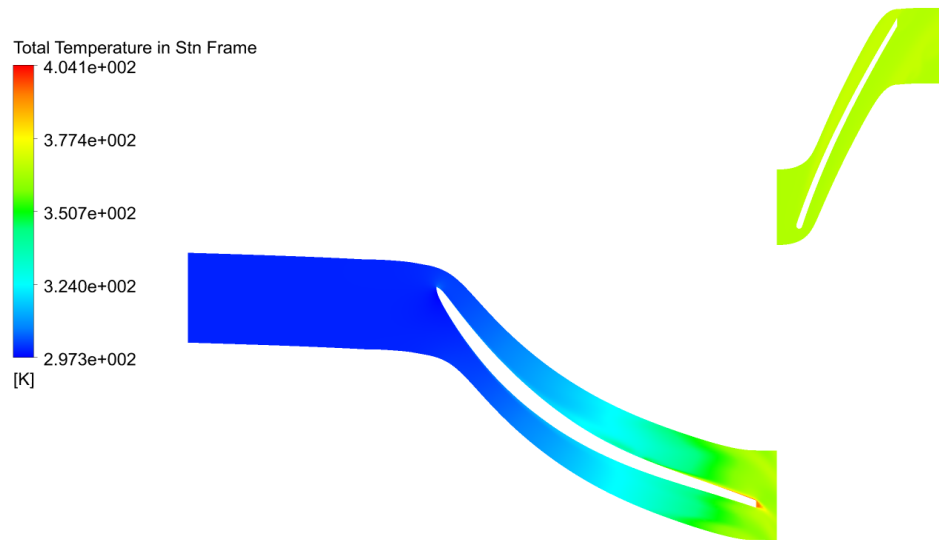


Figure 8. Total Temperature in Stn Frame.

On the impeller – diffuser interface, after the impeller blade, we can notice a slight pressure difference, given by the boundary layer detachment on the blade trailing edge, figure 9. Also, because the purpose of this component is to diffuse the flow, by decreasing the gas velocity; the pressure drop in the diffuser is normal - since there is intrinsic pressure losses associated with this process. The magnitudes of the pressure losses for the vanned diffuser are near 0.21bar.

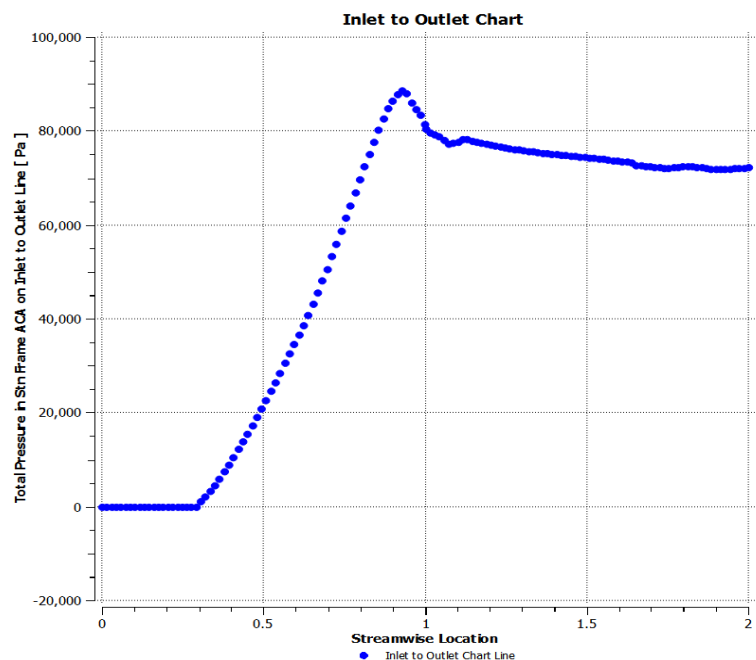


Figure 9. Inlet to outlet pressure distribution on the compressor.

Figure 10 presents the impeller blade loading at 50%span, showing the pressure variation on the blade surface. The blade pressure distribution is well balanced, showing only a small peak near the leading edge.

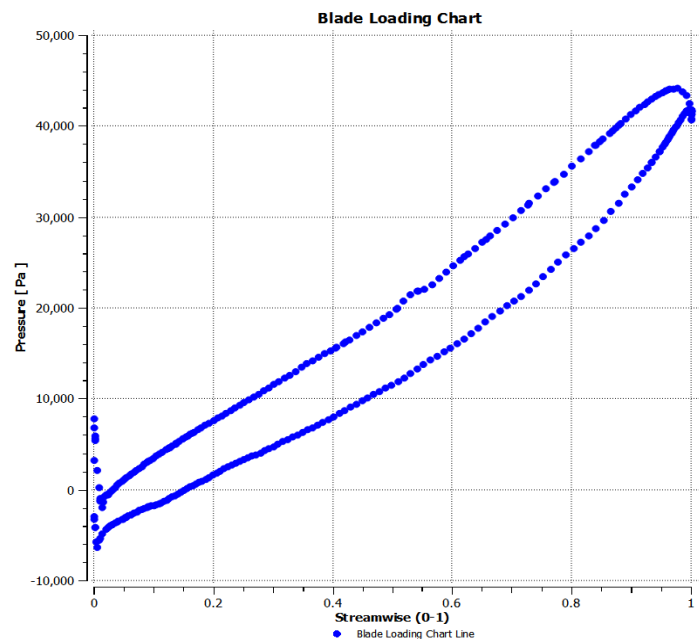


Figure 10. Impeller blade loading.

In figure 11 we show the locations in which the pressure probes have been installed. For CFD post-processing, these stream wise cuts were used for mass flow averaging the static and total pressures, while in the experimental setup individual measuring points were used. The first surface represents the vanned diffuser inlet, and the next one diffuser outlet.

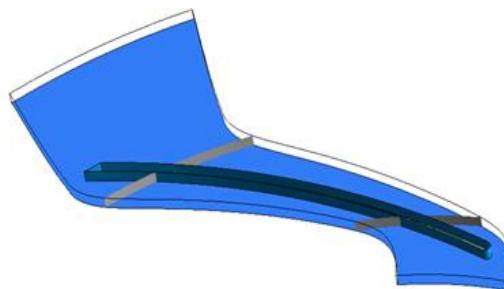


Figure 11. User surfaces in the measurement location.

Further is presented a comparison between experimental and numeric results (CFD analysis). Figure 12 illustrates the total pressure for the impeller and diffuser. The differences on the outlet sections are given by a difference of about 0.026 bars. For the vanned diffuser, the place where the total pressure is measured, are illustrated in Fig. 11. It is observed that between the two results, experimental and numerical, in the case of CFD analysis the results are higher, figure 12.b). This can be explained by the fact that some of the losses associated with the real stages that are ignored by the CFD model. On the outlet area a difference of about 0.14 bars for the total pressure is obtained. Experimental precision, on the other hand depends on the accuracy and the number of **the sensors used and number of runs and** their duration. For the static pressure, 12.e) pressure probes have been used in the experimental tests. The difference between the CFD and experimental can be presented in the terms of standard deviation, value of it being of about 0.077 bar.

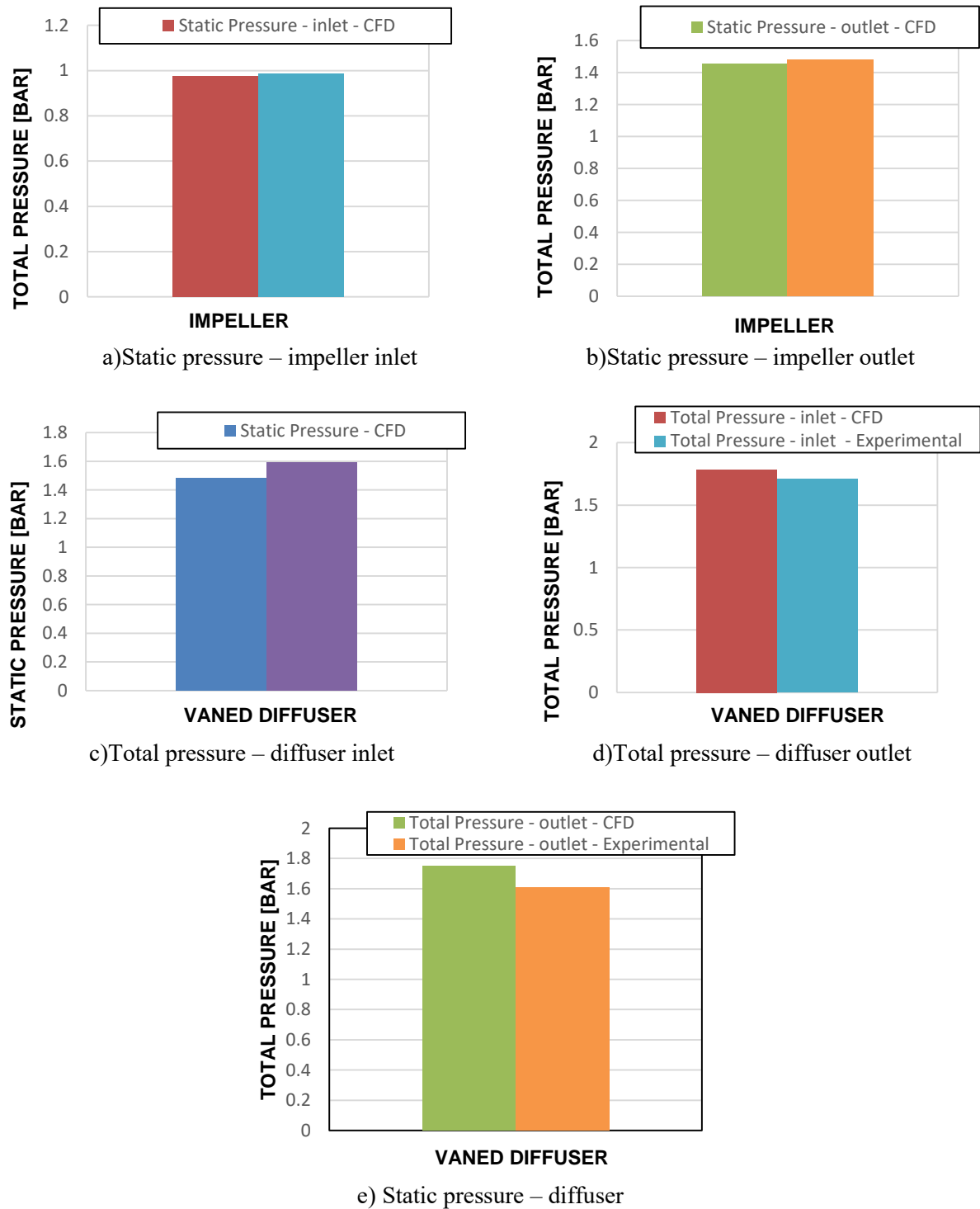


Figure 12. Comparison between experimental and CFD results.

4. Conclusion

This article compares centrifugal compressor performances measured on the test rig with the computational fluid dynamics results. Although the compressor was originally designed for operating on CH₄, the testing conditions used air, through the method of similitude. This comparison is made for

the similitude conditions; using as fluid flow ideal air. The test ring was set up and instrumented in accordance with the standard requirements, as described in the above sections. Overall, for both compressor components (impeller and diffuser) a number of 24 pressure probes were mounted. For the optimal design of a compressor stage, the impeller shape was improved based on CFD techniques; also the gas-dynamic calculation was performed on the model which is also optimized based on the stress analysis (finite element method).

Comparison showed that, differences between numerical and experimental are a result of working conditions variation. Higher differences between the pressures obtained are in the case of the vanned diffuser, due to difficulties in estimating diffusion losses. Although some differences have been observed, in terms of quantitative absolute values, the flow patterns have been appropriately determined qualitatively. Hence design optimization can be conceived and overall performance improved through this method.

5. References

- [1] Joost J and Brasz J 2006 Comparison of Part-Load Efficiency Characteristics of Screw and Centrifugal Compressors *International Compressor Engineering Conference*, Paper 1827, 1-2, West Lafayette, Indiana, United States, July 17-2
- [2] Denholm P, Erik E, Kirby B and Milligan M 2010 The Role of Energy Storage with Renewable Electricity Generation *Technical Report NREL/TP-6A2-4718* USA 35-46
- [3] El Merhubi H, Kéromnès A, Catalano G, Lefort B and Le Moyne L 2016 A high pressure experimental and numerical study of methane ignition *Fuel Elsevier* **177** 164–172
- [4] Wang Peng L, Cao Z, Yu B and Li W 2017 Similarity Conversion of Centrifugal Natural Gas Compressors Based on Predictor-Corrector *International Conference on Computational Science, ICCS 2017*, 12-14 June 2017, Zurich Switzerland
- [5] Galerkin Y, Mitrofanov V and Prokofiev A 2003 The Experience of CFD Calculations for Flow Analysis in Centrifugal Compressor Stages, TS – 020, *Proceedings of the International Gas Turbine Congress*, Tokyo, November 2-7 2003
- [6] Dixon S L, Hall C A 2010 *Fluid Mechanics and Thermodynamics of Turbomachinery Sixth Edition*, Elsevier Inc., ISBN 978-1-85617-793-1 218-258
- [7] Dragan V, Malael I and Gherman B 2015 Development of a Very High Pressure Ratio Single Stage Centrifugal Compressor *International Review on Modelling and Simulations (I.R.E.M.O.S.)* **8**(3)
- [8] Dragan V, Malael I and Gherman B 2016 A Comparative Analysis Between Optimized and Baseline High Pressure Compressor Stages *Engineering, Technology & Applied Science Research* **6**(4) 1103-1108
- [9] Dragan V 2017 Centrifugal compressor efficiency calculation with heat transfer, *IJUM Engineering Journal* **18**(2) 1-8
- [10] Bianchinia A, Carnevalea E A, Biliottib D, Altamoreb M, Cangemib E, Giachib M, Rubinob D T, Tapinassib L, Ferraraa G and Ferraric L 2015 Development of a research test rig for advanced analyses in centrifugal compressors *ATI 2015 - 70th Conference of the ATI Engineering Association Energy Procedia* **82** (2015) 230 – 236
- [11] Zemp A *CFD Investigation on Inlet Flow Distortion in a Centrifugal Compressor*, Swiss Federal Institute of Technology, ETH, Zurich, Master's Thesis 06/07
- [12] Sauss P Le, Fabrie P, Arnou D and Clunet F 2013 CFD comparison with centrifugal compressor measurements on a wide operating range, *EPJ Web of Conferences*, EDP Sciences, DOI: 10.1051/epjconf/20134501059
- [13] Smirnov P E and Menter F R 2009 Sensitization of the SST turbulence model to rotation and curvature by applying the Spalart-Shur correction term *ASME Journal of Turbomachinery* **131**(4) 041010 Jul 02, 2009
- [14] Georgiadis N J and Yoder D A 2006 Evaluation of Modified Two-Equation Turbulence Models for Jet Flow Predictions *AIAA Journal* **44**(12)