ISSN: 1231-4005
e-ISSN: 2354-0133
DOI: 10.5604/01.3001.0010.2902

MODERN METHODS OF IDENTIFICATION DESIGN CONDITIONS FOR SINGLE STAGE MICRO SCALE CENTRIFUGAL COMPRESSOR

Michał Czarnecki

Rzeszow University of Technology
Department of Aircraft and Aircraft Engines
Powstancow Warszawy Av. 8, 35-959 Rzeszow, Poland
tel.: +48 17 8651609, fax: +48 17 8543116
e-mail: czarn@prz.edu.pl

John Olsen

School of Mechanical and Manufacturing Engineering
The University of New South Wales, Sydney
Ainsworth Building, N.S.W., 2052, Sydney, Australia,
tel.: +61 2 93855217, fax: +61 2 96631222
e-mail: j.olsen@unsw.edu.au

Abstract

Micro scale gas turbine engines are low cost engines. They share their compressor impeller with automotive turbochargers. An identified design condition for the selected impeller is a critical stage of the design process. This process is had difficulties due the large number of manufacturers that provide OEM parts. It is common practice that one OEM part number provides the same impeller at different design revision. In general, parts are interchangeable but in detail, they differ slightly in terms of dimensions and performance. To avoid under predict or over predict inputs data, it is important to check the design parameters with as many methods as possible. In practice, the designer could rely on analytical methods, which are straightforward limited to the applied design. When shared its (compressor operation) it is recommended additional information be provided by computational fluid dynamics that produces a three-dimensional look into the predesign. That allows avoidance of future design failure and reduces both design time and prototype manufacturing costs.

Keywords: *jet engine, gas turbine, design, off design, trend prediction, centrifugal compressor*

1. Introduction

This article presents a part of a micro scale, gas turbine design process. The process begins from the selection of a centrifugal compressor impeller from available automotive parts. After studying compressor maps, the K24 compressor was chosen [6]. During the test of first engine, *m*SO-1, the centrifugal compressor was damaged due an insufficient rotor preloading force. A replaceable compressor wheel was purchased. In comparing both parts with the same OEM number 5324-123-2017, a difference in the dimensions was found (Tab. 1 – Part no 1 – initial design, Part no 2 – improvement design).

In comparing different vendors, it is become clear that previous design conditions cannot be trusted. Therefore, an additional study of the compressor performance map was made. For the K24 compressor design three different compressor maps were found: type 2464 – 0.165 kg/s [7], type 2467 – 0.165 kg/s [8], and type 2470 – 0.19 kg/s [9]. The difference between the 2464 and the 2467 design is the efficiency island distribution on the compressor performance.

Using analytical approaches, the mass flow rate was estimated at a rotational speed of n = 120,000 r/min for given compressor wheel. For two analytical methods, the estimated mass flow rate were 0.1465 kg/s and 0.1799 kg/s [3].A straightforward mass flow rate comparison was impossible due to the difference in diffuser design of the turbocharger stage and jet engine compressor. There is an assumption that an influence between the rotor and stator in operation, affects the flow.

Tab. 1. KKK K24 compressor wheel revisions

OEM / PN	Inducer diameter	Exducer diameter	Shaft diameter	Blade main/ /splitter	Bottom type	Manufacturer
5324-123-2017 4COW-3860N-0100	38.8	60.6	7.00	6/6	flatback	ZAGE
5324-123-2000	36.2	60.6	7.00	6/6	flatback	ZAGE
5324-123-2014	36.2	60.6	7.00	6/6	flatback	ZAGE
5324-123-2017 4COW-3660N-0100	36.2	60.6	7.00	6/6	flatback	ZAGE
4COW-4768N-0100	47.5	68.5	7.00	7/7	superback	ZAGE
5324-970-7400	45.70	65.30	5.40	6/6	superback	OEM
	54			6/6	superback	Ultimate Motorwerks
5324-123-2017	38.6	60.2	5.40	6/6	flatback	Part no 1
5324-123-2017	38.8	60.5	7.00	6.6	flatback	Part no 2

To avoid under or over prediction of the estimated value, additional steps were taken into account.

For precise mass flow rate estimation, an additional method was proposed. This method combines modern engineering tools – CAD (CATIA V5) and CFD (ANSYS CFX). The idea of obtaining additional information was to reverse engineer the compressor geometry, and then transfer this geometry into CAD environment and finally estimate of a mass flow rate by from the CFD results.

2. Identification procedure

To develop a proper procedure for calculating the compressor stage performance, it is essential to identify limitations within the numerical environment. First limitation is related to the software license limitation, which is "ANSYS Academic Research". The limitation for number of nodes for that license is 512k for any of the CFD solvers. If a model is built that is larger than 512k, the message shown in (Fig. 1) is displayed.

```
+------------------------------------------------------------------+
| ERROR #001100247 has occurred in subroutine .                    |
| Message:                                                         |
|                                                                  |
| The solver is unable to continue because of licensing problems.  |
|                                                                  |
| A license for the following capability level could not be checked |
| out:                                                             |
|                                                                  |
| ANSYS CFX Solver (> 512K Nodes) + Parallel                       |
|                                                                  |
```

Fig. 1. Numerical model size limitation up to 512K nodes

The second limitation is introduced due the conceptual character of calculations. This limitation is related to the time of computation. At the conceptual stage, the total calculation time at the design point should not be greater than 1 hour. This limit is a consequence of the large number of calculations at different design variants and on calculations at design and in off design conditions. For off design conditions, the numerical model is tested at the end of the range of applicability, and then a time to get numerical convergence is increased by between 3 to 5 times. The developer of ANSYS software recommends that at the conceptual stage, the total computational time should be reduced up to 2 hours.

The third limitation is related to total computational power of the cluster that is dedicated to calculate the various cases. For these calculations, the dedicated node of UNSW Leonardi Cluster was selected. A single node of a cluster consists of 2.4-2.3 GHz Opteron 6174 CPU, 96 GB or RAM.

The fourth limitation is related to meshing and geometry setup. The geometry is prepared in the CATIA V5 environment and then converted and meshed in the Design Modeller in ANSYS. This is a compromise between time and efficiency. The compressor rotor geometry is reverse engineered, (inlet and diffuser was designed at CATIA) as a result of implementing estimated values from the analytical model. Additional work must be done due possible errors in geometry translation.

For numerical simulation of the compressor centrifugal stage, the calculation procedure was introduced (Fig. 2). The calculation was divided into two stages.

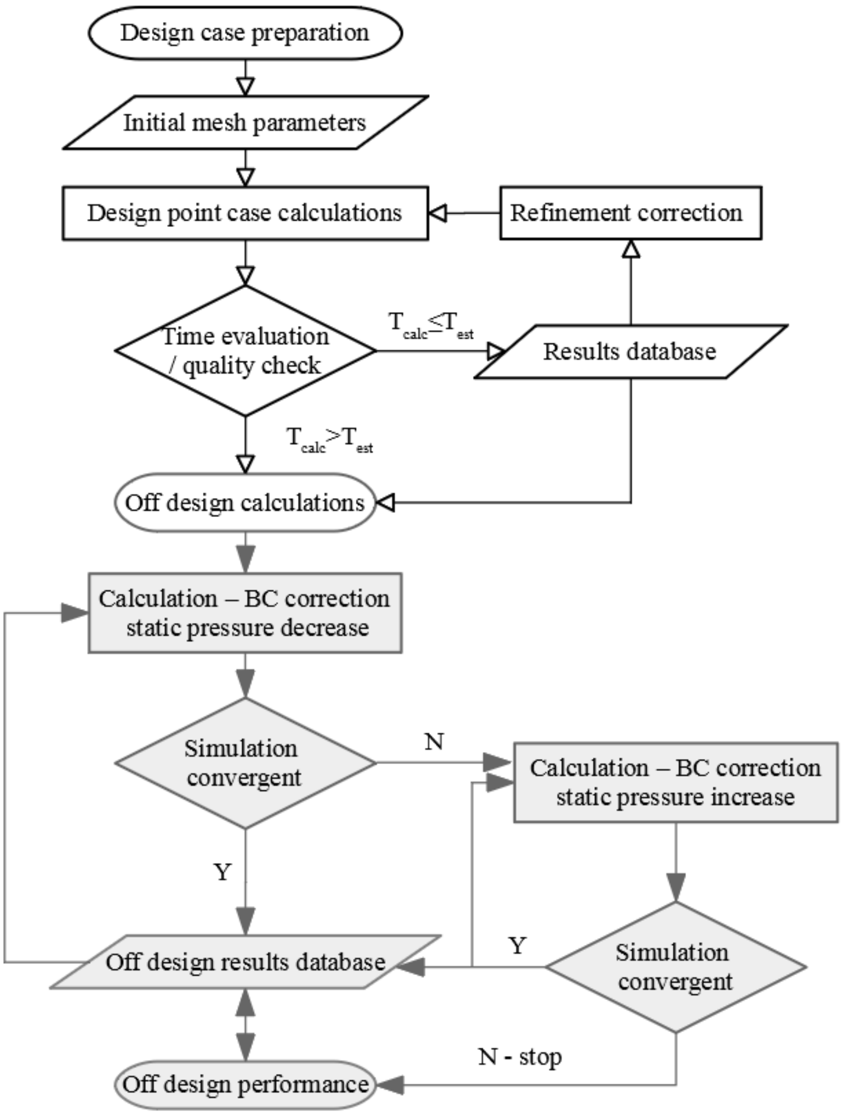

Fig. 2. Design and off design procedure for centrifugal compressor stage (white part – design point calculations, grey off design performance for selected velocity line)

The first stage consists of numerical calculations at the design point condition. The calculations were initially performed for a case that consisted of a small number of nodes. This simple model was chosen for testing predefined boundary conditions and the turbulence model. When these conditions were properly established, the number of nodes was gradually increased. The first stages of calculations were finished when the computational calculation time T_{calc} became greater than the estimated computational time T_{est}, set at 60 min. From that point on, the number of nodes and cells is locked.

The next stage consisted of off-design conditions. For selected performance map (constant) rotational speed curve, off-design conditions are tested. Selected rotational speed curve chosen is n = 120,000 r/min which corresponds to design conditions [3]. This allows the prediction of stage behaviour and gives an opportunity to compare with the compressor stage from turbocharger design. In the aero industry, off design calculations are referred to as “throttling procedure”). Boundary conditions for inlet are similar to the initial values. The variables are only the static pressure at the diffuser exit. At first, the static pressure is decreased to identify maximum airflow capacity. When convergence cannot be reached, calculations are stopped. Conditions are again set up do analytical design point (ADP) values. From that point, the static pressure at the exit from the stage is increased until convergence again becomes unstable.

3. Model and mesh setup

The compressor wheel model setup was divided into three parts:
- preparing the compressor wheel for the scanning procedure by cleaning and adding reference points (Fig. 3a),
- converting the compressor geometry into a cloud of points (Fig. 3b),
- transferring the geometry into a solid model (Fig. 3c),

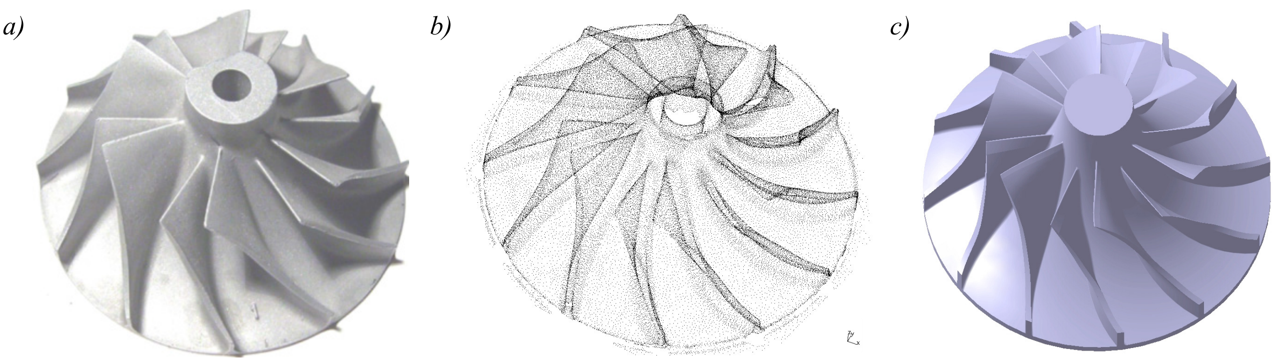

Fig. 3. K24 compressor wheel geometry. a) real design, b) cloud of points, c) solid model

Fluid domain model consists three subdomains, namely: the inlet subdomain, the rotor subdomain and the stator subdomain (Fig. 4).

For the semi-automated Design Modeller meshing procedure, additional setup preferences were selected:
- in “Defaults” menu:
  - Physics preference: CFD,
  - Solver preference: Fluent,
- in “Sizing” menu:
  - Use Advanced Size Function: On Curvature,
  - Relevance Center: Fine,
  - Smoothing: Medium,
  - Transition: Slow,
  - Span Angle Center: Fine.

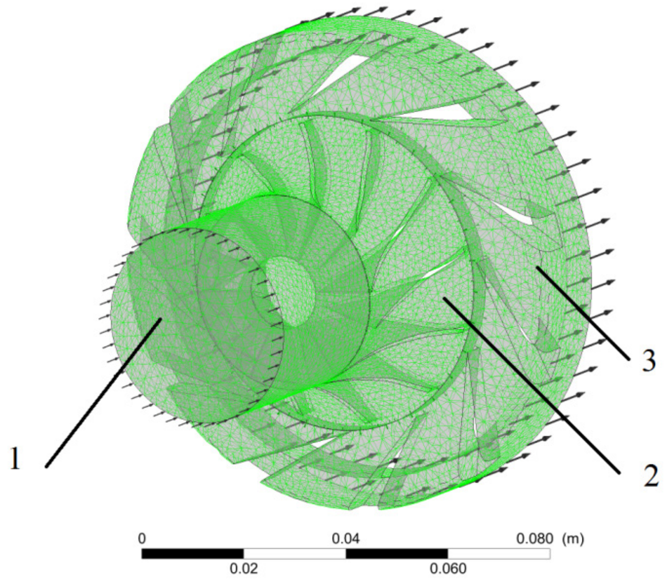

Fig. 4. Centrifugal compressor stage domain. 1) intake subdomain, 2) rotor subdomain, 3) stator subdomain

For the selected time calculation limitations, additional setup in "Span Angle Center" was required. "Min size" was set to 0.4 mm, while Max Face Size was set to 0.6 mm and Max Size set to 1.2 mm. Leaving cells size meshed in "predefined size" lead to unproportioned mesh size between inlet, rotor and stator, which is a cause of numerical inaccuracy.

The boundary conditions are:
- selected rotational speed for impeller domain,
- constant total pressure at the intake,
- variable static pressure at compressor exit,
- coupling method between rotational and non-rotational parts – stage,
- advection scheme – first order / upwind,
- convergence criteria – RMS value 0.00001.

4. Design point model size estimation

To estimate the domain size that can be calculated at less than 1 CPU hour, eight cases are considered (Tab. 2). The size for a cluster node should be 400 up to 450 nodes, as that size corresponds to the license limitations (Fig. 1). A semi automation meshing procedure is recommended to generate the CFD domain at 400-node range due the partially random process that influences the final number of nodes.

Tab. 2. Numerical model size estimation

Test case	Total value		Intake		Rotor		Stator	
	Nodes	Elements	Elements	Nodes	Elements	Nodes	Elements	Nodes
No. 1	22762	83331	38272	9523	8297	28839	16220	4942
No. 2	35098	142513	39923	8407	11256	42898	59692	15435
No. 3	85493	359216	68075	15302	169026	37788	122115	32403
No. 4	159053	718945	75821	17976	182767	45941	460357	95136
No. 5	169665	751438	149553	32084	371869	79002	230036	58579
No. 6	196335	940401	189430	38079	311157	66994	439814	91262
No. 7	260913	1297440	214016	41980	623067	123797	460357	95136
No. 8	438221	2193929	419795	80725	841233	165615	975598	193130

The minimum number of nodes for the design study should be greater than 150 nodes due to total pressure and efficiency estimation (Tab. 2, 3).

Tab. 3. Numerical calculations result at ADP

Parameter	m	p_{04}	η_c
Test case / Unit	[kg/s]	[bar]	[%]
No. 1	0.16313402	2.109224	66.524
No 2	0.165762226	2.153008	68.253
No. 3	0.1649038	2.1424	69.154
No 4	0.16587966	2.179112	69.698
No 5	0.16659088	2.163824	69.716
No. 6	0.1664751	2.182024	70.732
No. 7	0.16809602	2.195648	69.886
No. 8	0.16641	2.189678	70.658

In terms of maximum estimation time, the limited restriction of the number of computational nodes does not allow us to exceed total computational given time (Fig. 5).

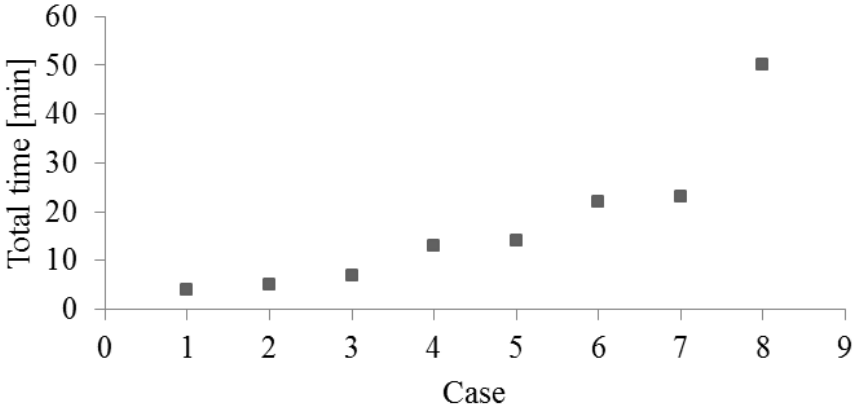

Fig. 5. Numerical model total calculation time for design condition

Comparing ADP– (see Tab. 3) to the computational fluid dynamics simulation, the mass flow rate numerical result is under predicted by 3.6%. The total pressure is over predicted by 2%, while the overall isentropic efficiency is under predicted by 3.2%. The total power required to drive compressor is under predicted by 0.7%. To provide a safety margin for under and over predictions by different calculation methods, it is necessary to apply a safety factor for power generation. Multiplying the estimated power required to make the compressor stage operational by 1.02 factor makes the design in terms of the mathematical and numerical model workable in both design and off design conditions. That allows us to rely on analytical model only.

5. Off design performance prediction

Next part of a calculation corresponds to off design performance of the compressor stage at predefined rotational speed – 120,000 r/min. Numerical simulations were compared with two approaches to the modelling the turbulence – advection scheme: high resolution and upwind. The upwind procedure is recommended for off design condition calculations for centrifugal compressor performance estimation [1, 5].

The numerical results were compared to a K24 turbocharger with impeller compatible with reverse engineered one. For comparison, two rotational speed lines from performance maps with impeller K24-2464 and K24-2470 were selected [7, 9]. These data are only for general overview because automotive industry often upgrades their designs without detailed notification to the market, but only by a "compatible" note. The "high resolution" pressure calculation is sensitive to static pressure conditions. When the maximum pressure is archived, the model becomes highly unstable, and a slight difference in pressure generates a large penalty in airflow calculation (Fig. 6). the "upwind" advection scheme in terms of quality of performance line, are comparable with automotive design. The aero design with dedicated stator blades archives higher-pressure ratios with stable operation range penalty. The advantage of aero diffuser over an automotive diffuser is that the total pressure ratio is up to 7%.

In terms of isentropic efficiency, the aero engine diffuser is less effective due a shorter and directed diffuser channel (Fig. 7) compared to the automotive solution. The isentropic efficiency penalty at off design conditions is up to 7.6 %

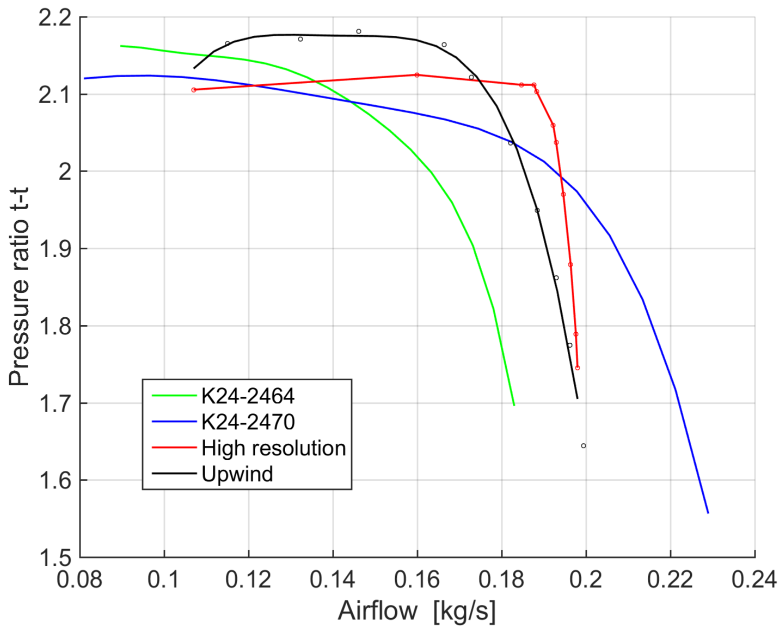

Fig. 6. Pressure ratio estimation

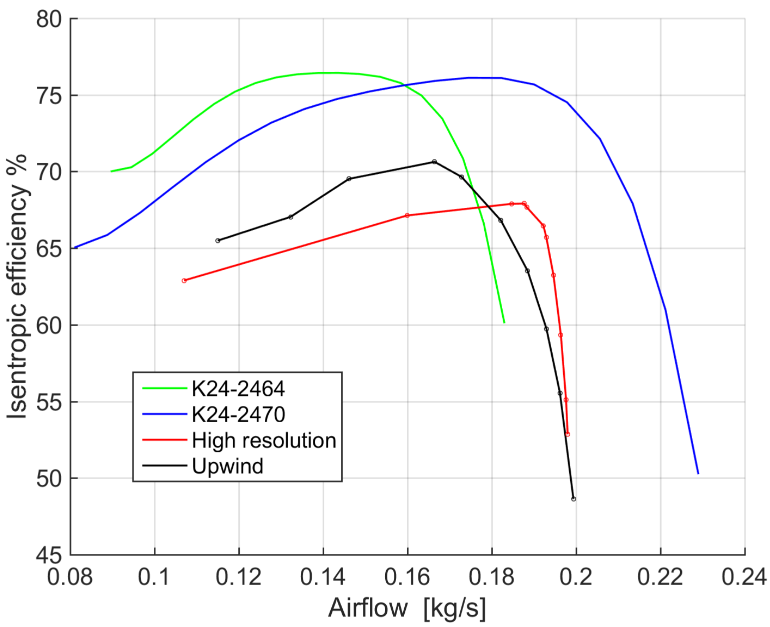

Fig. 7. Isentropic efficiency estimation

6. Design point CFD results

At ADP fluid flow qualitative analysis figure scales are predefined as follows:

- for static pressure ratio scale is from 1 up to 2.5 [bar],
- for total pressure ratio scale is from 1 up to 3 [bar],
- for velocity scale is set from 0 to 300 or 400 [m/s],
- for total temperature scale is set up from 280 to 400 [K],
- for turbulence and kinetic energy scale range from 15 to 800 [m^2/s^2] is defined.

For meridional plane total and static pressure, distribution is presented below.

The obtained result in terms of numerical correctness to parameter distribution presented in theory is consistent in quality [2, 4] (Fig. 8).

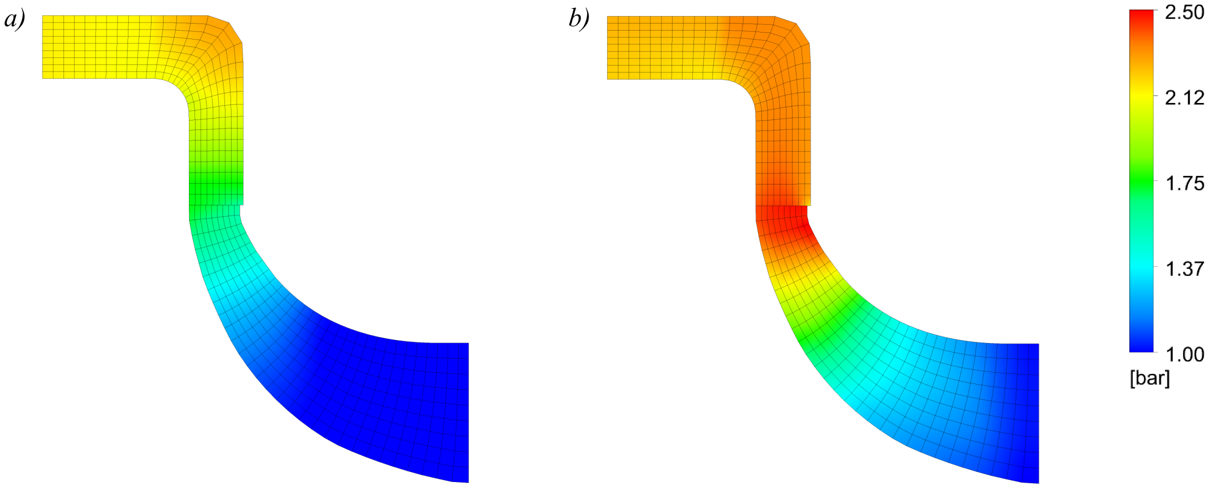

Fig. 8. Pressure ratio estimation

The temperature increment in the compressor stage is consistent in quality to the theoretical results [2, 4]. For a given pressure ratio difference, the average total temperature at the exit of the compressor is 389.8 K.

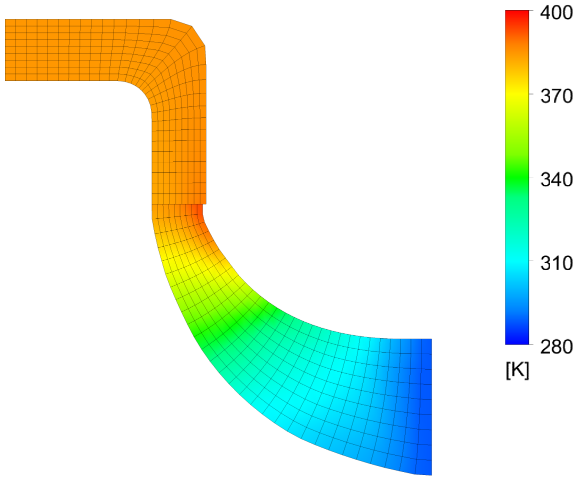

Fig. 9. Pressure ratio estimation

Evaluation of a fluid flow through stator should be verified up to the 90° turn. The turned channel should be treated as a feed channel to a combustor (Fig. 10a). That is the result of integrating combustor feed channel into micro scale diffuser structure. Qualitative verification of a level of turbulence is required to evaluate the turbulence intensity spots (Fig. 10b). A high level of turbulence decreases the overall thermodynamic efficiency in a flow channel. Sources of turbulence are located at the compressor inlet tip due high Mach number ~1. Another spot is located at stator rotor connection. The turbulence intensity at this spot is an effect of the stator – rotor interaction. The different proportions between hub and shroud of a compressor are due to the effect of centrifugal forces on fluid flow. The sharp gap between the rotor and stator are effect of technological tolerance required for the compressor to operate.

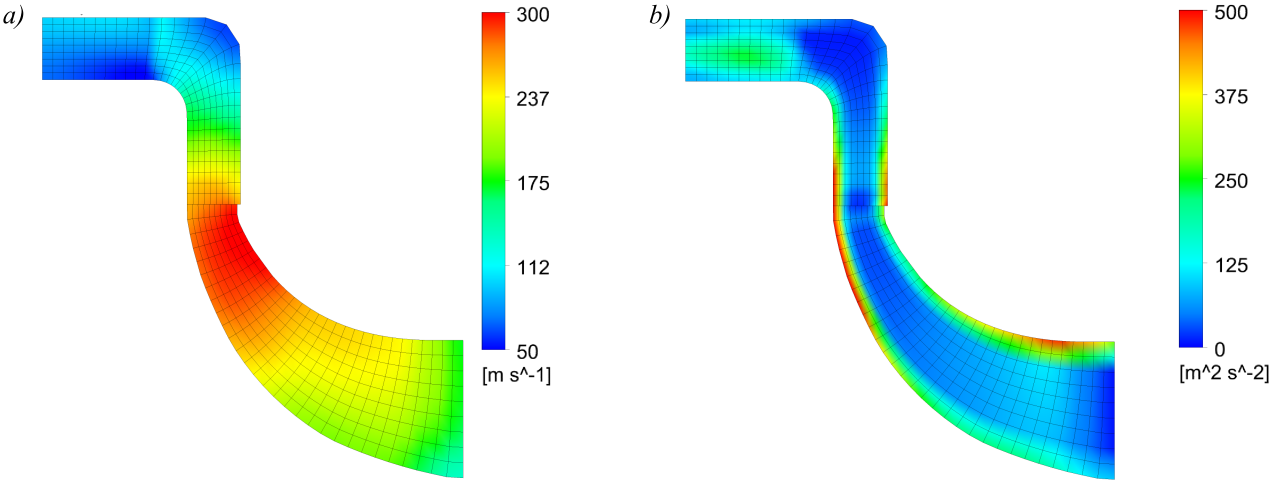

Fig. 10. a) Velocity distribution at meridional plane, b) turbulence and kinetic energy

Blade to blade surface was also setup for different proportion of tip/shroud height. Their ratios are 0.25, 0.5, 0.75 (Fig. 11).

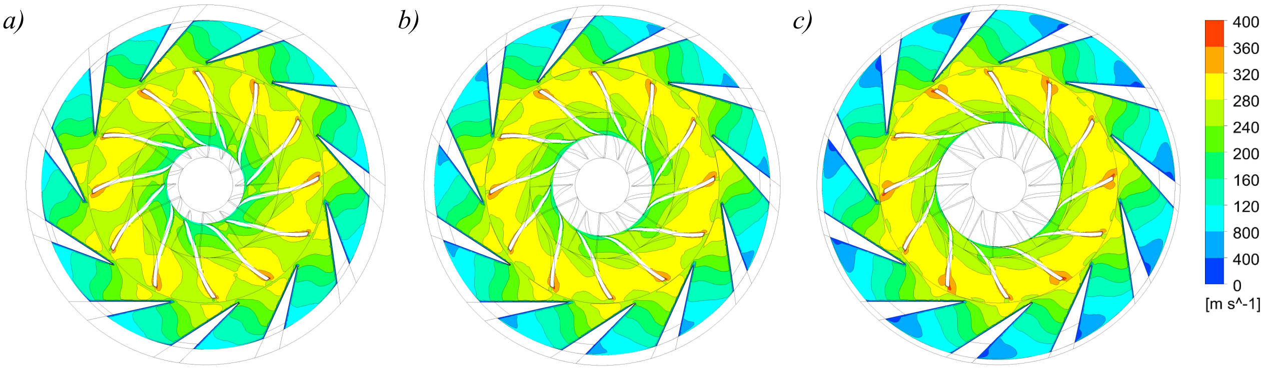

Fig. 11. Velocity vectors: a) blade to blade 25%, b) blade to blade 50%, c) blade to blade 75%

The distribution of velocity in a stage is correct when blade to blade ratio is increased; the velocity in the rotor is increased and the velocity in stator is decreased, due the strong effect of centrifugal forces on a fluid flow.

The total pressure distribution is correct, while the rotor to stator interaction is also correct as the velocity profile that corresponds to analytical model.

To proper evaluate, the diffuser it is required to split and individually evaluate the stator to focus on its efficiency because the inlet and rotor is a unified design. The criteria for the evaluation of the centrifugal diffuser stage are:

– velocity, that allows to check profile of isolines and decrease of velocity in the channel that is proportional to pressure recovery,
– turbulence level that is required to evaluate efficiency of design through friction in the flow channel,
– total pressure – important to evaluate the real design by measurement of the total pressure.

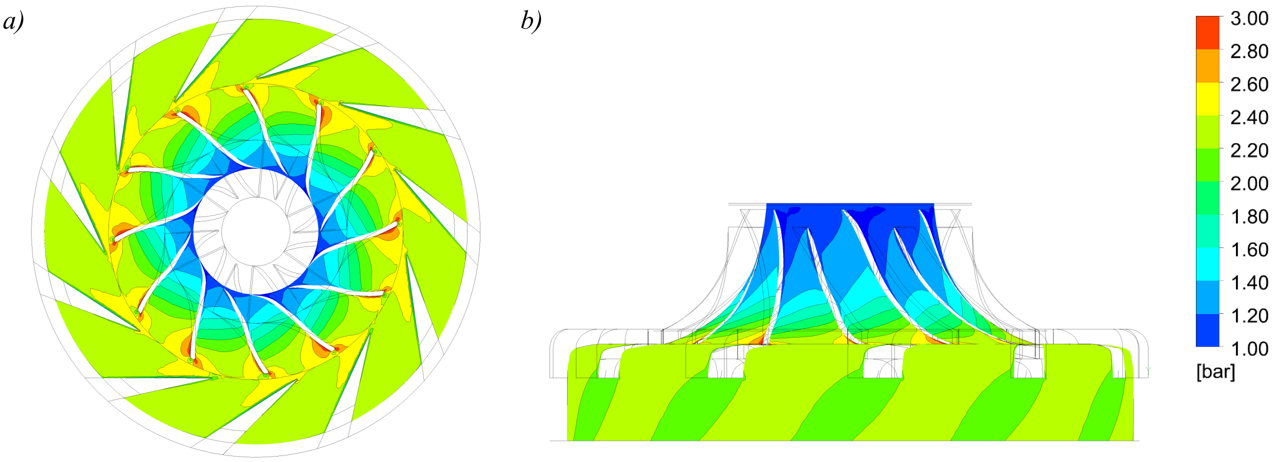

Fig. 12. Total pressure: a) front view, b) side view

The velocity profile in basic design diffuser has a subsonic character. The profile of the isolines decreases as the diameter of the diffuser increases. The wedge profile vanes are responsible for uneven distribution of velocity profile (Fig. 13b). The recommended diffuser angle is 12° but that is impossible for that engine layout. Due to the downscaling of the engine diameter in comparison do KJ-66, the diffuser angle is 14.5° due assembly connections between compressor case and diffuser body (M3 – type thread).

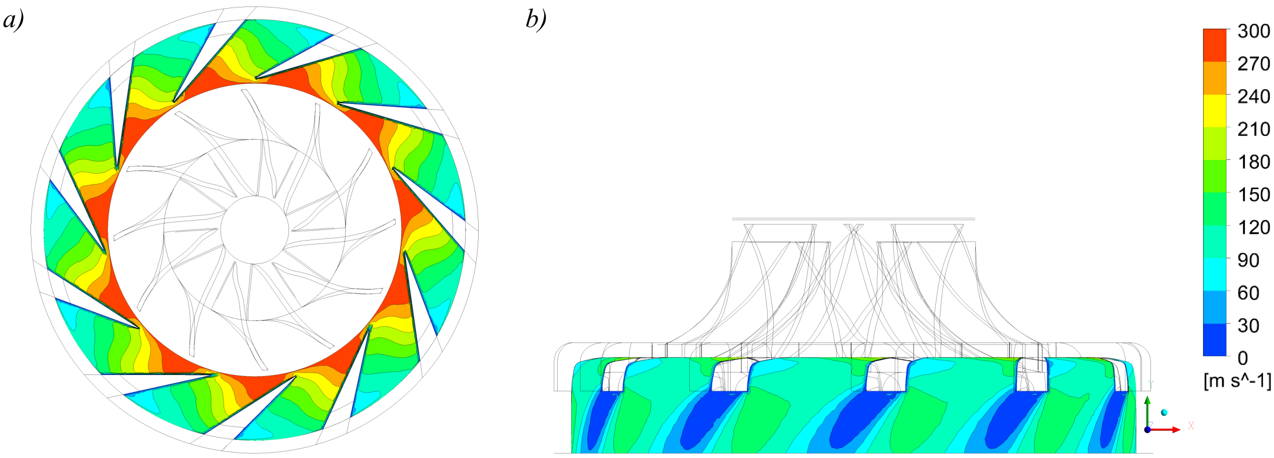

Fig. 13. Velocity: a) front view, b) side view

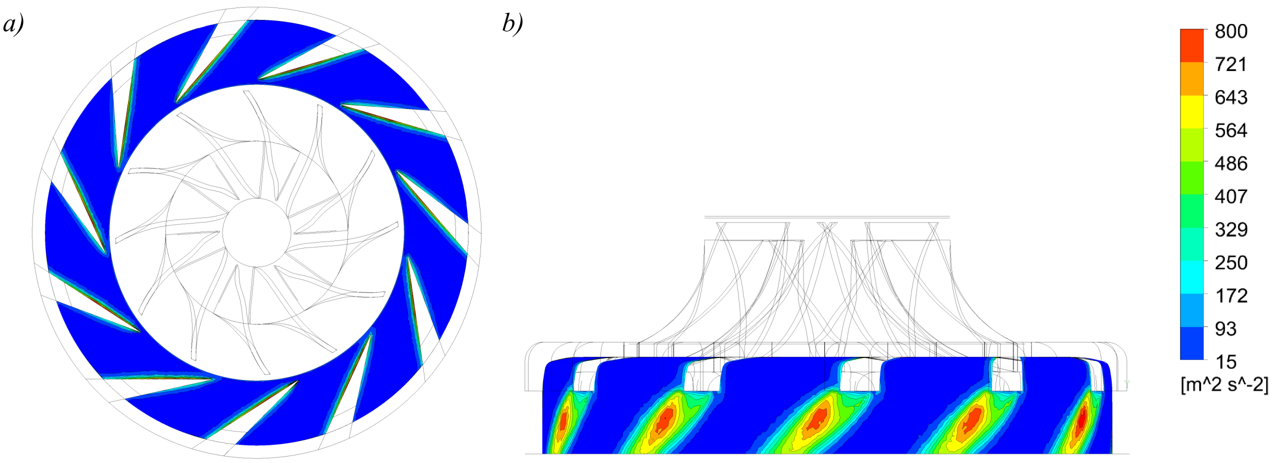

Fig. 14. Turbulence kinetic energy: a) front view, b) side view

In terms of turbulence in the diffuser channel, there is a slight difference between pressure and suction side of a diffuser blade. The disadvantage of a wedge bladed diffuser is the strong turbulence at the exit of a diffuser channel in the low speed region (Fig. 14b). The distribution of total pressure in diffuser channel in terms of quality is correct [4]. The benefit of the bladed diffuser is a higher static pressure recovery at the exit (Fig. 15) due a longer flow channel with wider area.

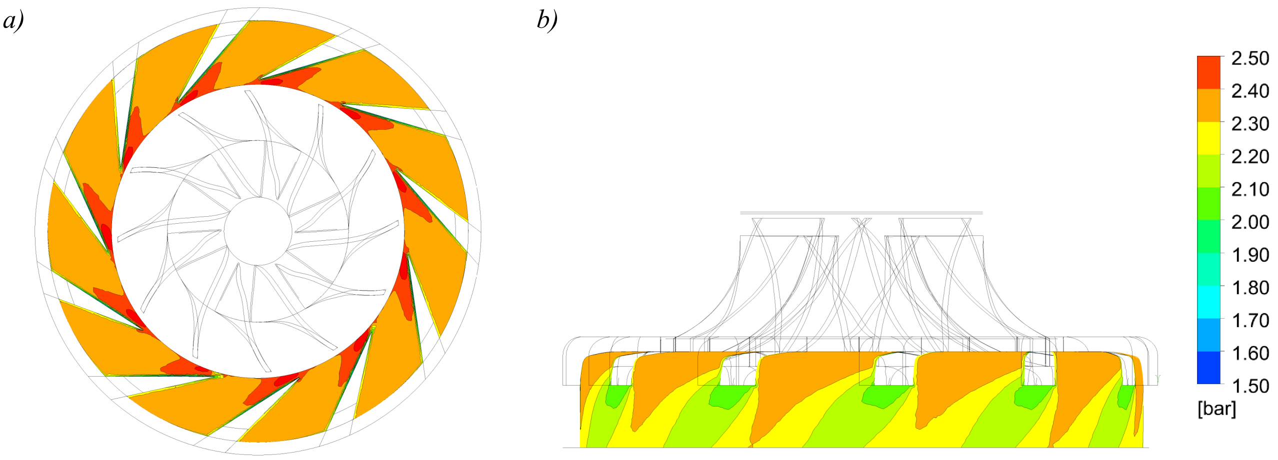

Fig. 15. Total pressure profile: a) front view, b) side view

7. Summary

The obtained results show critical areas in diffuser design. There are two main areas, which are important to compressor operation:

– vaneless diffuser,
– exit from vane part of diffuser

For the vaneless diffuser it is important to operate in the subsonic speed range. At the exit of the diffuser, the designer should provide a low speed profile with maximum pressure recovery (requirements for combustion process). Whole process of reverse engineering needs to be carried carefully due a high risk of numerical error in one of two crucial part, namely, geometry translation and meshing procedure [3].

Applied CFD methods allow us to identify possible impeller types. The identified designs are similar in terms of mass flow rate. In comparison to the analytical solution, the CFD prediction offers clear answer for a given question (Fig. 16). The averaged analytical results also match compressor data; however, without a numerical solution at this scale of detail, it was impossible to rely only on theory.

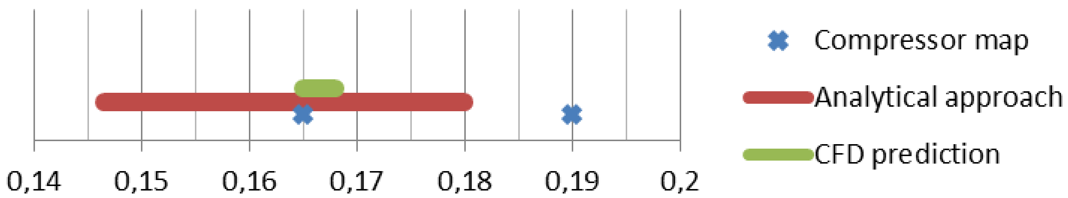

Fig. 16. Mass flow rate prediction

Acknowledgements

Presented work was an effect of a PANTHER research project, which was awarded under Erasmus Mundus Action II by EU.

References

[1] Avaramenko, A., Frolov, A., Hamalainen, J., *Simulation and modelling of flow in gas compressor*, Journal of Applied Mathematics, Volume 2015.

[2] Cohen, H., Rodgers, G. F. C., Saravanamutto, H. I. H., *Gas turbine theory,* Longman Group limited, Essex, 1996.

[3] Czarnecki, M, *Micro scale gas turbine diffuser design – a practical approach,* Journal of Applied Fluid Mechanics

[4] Dixon, S. L., Hall, C. A., *Fluid mechanics and thermodynamics of turbomachinery*, Elsevier, Tokyo 2010.

[5] Tu, J. Y., Yeoh, G. H., Liu, C, *Computational Fluid Dynamics: A practical approach*, Elsevier, 2012.

[6] BW*, Performance turbocharger catalog 2012 Edition* (BorgWarner Turbo Systems).

[7] KKK Verdichterkennfeld 2464.

[8] KKK Verdichterkennfeld 2467.

[9] KKK Verdichterkennfeld 2470.